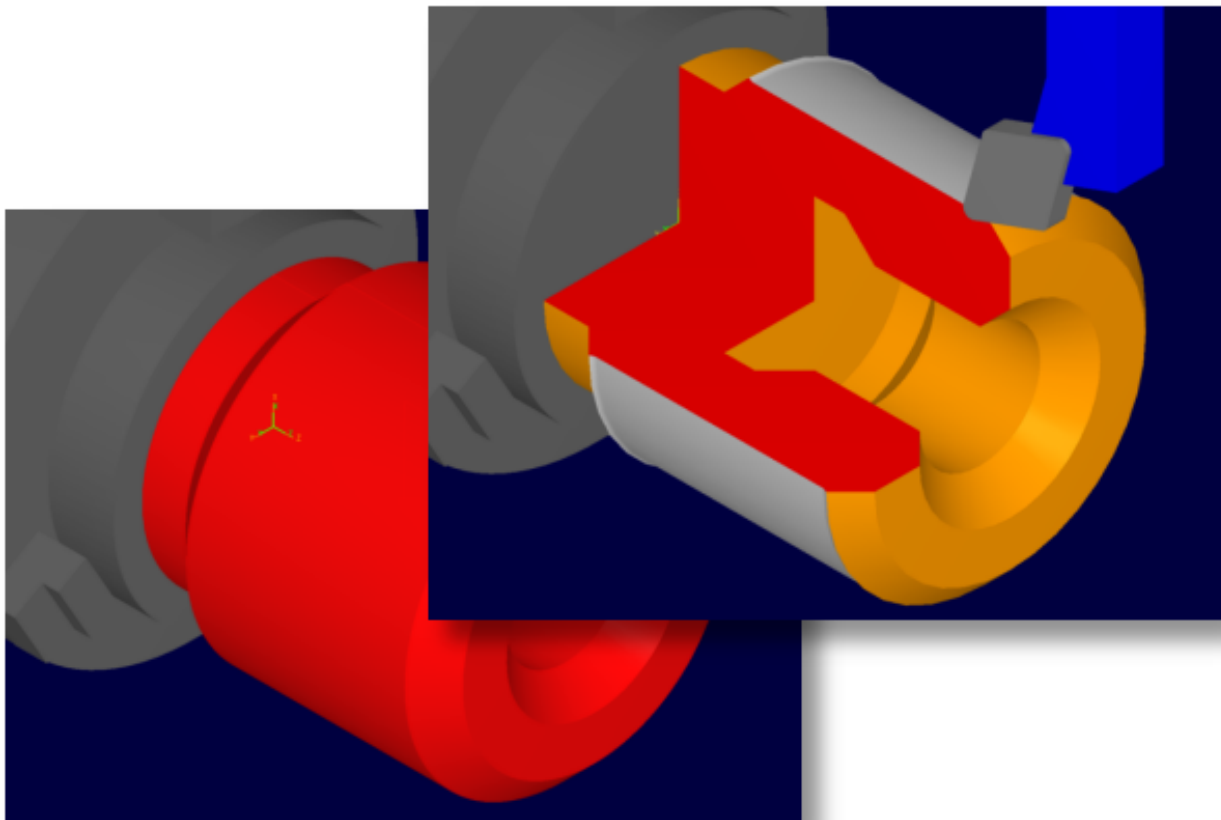


VisualTurn®



Easy to use 2-axis lathe programming system



MecSoft Corporation

End-User Software License Agreement

This MecSoft Corporation's VisualTurn End User Software License Agreement that accompanies the VisualTurn(TM) software product ("Software") and related documentation ("Documentation"). The term "Software" shall also include any upgrades, modified versions or updates of the Software licensed to you by MecSoft.

MecSoft Corporation grants to you a nonexclusive license to use the Software and Documentation, provided that you agree to the following:

1. USE OF THE SOFTWARE.

You may install the copy on multiple computers. You may not have more than the legally purchased number of licenses of Software running concurrently at one time.

2. COPYRIGHT.

The Software is owned by MecSoft Corporation and its suppliers. The Software's structure, organization and code are the valuable trade secrets of MecSoft Corporation and its suppliers. The Software is also protected by United States Copyright Law and International Treaty provisions. You must treat the Software just as you would any other copyrighted material, such as a book. You may not copy the Software or the Documentation, except as set forth in the "Use of the Software" section. Any copies that you are permitted to make pursuant to this Agreement must contain the same copyright and other proprietary notices that appear on or in the Software. You agree not to modify, adapt, translate, reverse engineer, de-compile, disassemble or otherwise attempt to discover the source code of the Software. Trademarks shall be used in accordance with accepted trademark practice, including identification of trademark owner's name.

Trademarks can only be used to identify printed output produced by the Software. Such use of any trademark does not give you any rights of ownership in that trademark. Except as stated above, this Agreement does not grant you any intellectual property rights in the Software.

3. TRANSFER.

You may not rent, lease, sublicense or lend the Software or Documentation.

4. LIMITED WARRANTY.

MecSoft Corporation warrants to you that the Software will perform substantially in accordance with the Documentation for the thirty (30) day period following your receipt of the Software. To make a warranty claim, you must notify MecSoft Corporation within such thirty (30) day period. If the Software does not perform substantially in accordance with the Documentation, the entire and exclusive liability and remedy shall be limited to either the replacement of the Software or the refund of the license fee you paid for the Software.

MECSOFT CORPORATION AND ITS SUPPLIERS DO NOT AND CANNOT WARRANT THE PERFORMANCE OR RESULTS YOU MAY OBTAIN BY USING THE SOFTWARE. THE FOREGOING STATES THE SOLE AND EXCLUSIVE REMEDIES FOR MECOFT CORPORATION'S OR ITS SUPPLIERS' BREACH OF WARRANTY. EXCEPT FOR THE FOREGOING LIMITED WARRANTY, MECOFT CORPORATION AND ITS SUPPLIERS MAKE NO WARRANTIES, EXPRESS OR IMPLIED, AS TO THE NON-INFRINGEMENT OF THIRD PARTY RIGHTS, MECHANABILITY, OR FITNESS FOR ANY PARTICULAR PURPOSE. IN NO EVENT WILL MECOFT CORPORATION OR ITS SUPPLIERS BE LIABLE TO YOU FOR ANY CONSEQUENTIAL, INCIDENTAL OR SPECIAL DAMAGES, INCLUDING ANY LOST PROFITS OR LOST SAVINGS, EVEN IF A MECOFT CORPORATION REPRESENTATIVE

Getting Started with VisualTurn

HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES OR FOR ANY CLAIM BY ANY THIRD PARTY.

Some states or jurisdictions do not allow the exclusion or limitation of incidental, consequential or special damages, or the exclusion of implied warranties or limitations on how long an implied warranty may last, so the above limitations may not apply to you. To the extent permissible, any implied warranties are limited to thirty (30) days. This warranty gives you specific legal rights. You may have other rights which vary from state to state or jurisdiction to jurisdiction. For further warranty information, please contact MecSoft Corporation's Customer Support.

5. GOVERNING LAW AND GOVERNING PROVISIONS.

This Agreement will be governed by the laws in force in the State of California excluding the application of its conflicts of law rules. This Agreement will not be governed by the United Nations Convention on Contracts for the International Sale of Goods, the application of which is expressly excluded. If any part of this Agreement is found void and unenforceable, it will not affect the validity of the balance of the Agreement, which shall remain valid and enforceable according to its terms. You agree that the Software will not be shipped, transferred or exported into any country or used in any manner prohibited by the United States Export Administration Act or any other export laws, restrictions or regulations. This Agreement shall automatically terminate upon failure by you to comply with its terms. This Agreement may only be modified in writing signed by an authorized officer of MecSoft Corporation.

6. U.S. GOVERNMENT RESTRICTED RIGHTS

Use, duplication, or disclosure by the government is subject to restrictions as set forth in subparagraph (c) (1) (ii) of The Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 or subparagraphs (c) (1) and (2) of Commercial Computer Software – Restricted Rights at 48 CFR 52.227-19, as applicable. Manufacturer is: MecSoft Corporation, 18019, Sky Park Circle, Suite KL, Irvine CA – 92614-6386, USA.

Unpublished - rights reserved under the copyright laws of the United States.

MecSoft Corporation
18019, Sky Park Circle, Suite KL
Irvine, CA 92614-6386

VisualTurn is a registered trademark of MecSoft Corporation
© 1998-2006+, MecSoft Corporation

Trademark credits

Windows is a registered trademark of Microsoft Corporation
Pentium is a registered trademark of Intel Corporation
Rhino is a registered trademark of McNeel & Associates.

Table of Contents

WELCOME TO VISUALTURN.....	5
ABOUT THIS GUIDE	5
COMPUTER REQUIREMENTS	5
INSTALLING VISUALTURN	6
RUNNING VISUALTURN	9
VISUALTURN USER INTERFACE.....	10
VISUALTURN BROWSER WINDOW	11
VISUALTURN TOOLBARS	11
VISUALTURN WORKFLOW.....	12
TYPICAL SCENARIO.....	13
PROGRAMMING WORKFLOW	13
POST-PROCESSING	14
MACHINING METHODS	15
TURNING OPERATIONS.....	15
HOLE-MAKING OPERATIONS	20
KEY CONCEPTS IN VISUALTURN PROGRAMMING.....	22
TURNING COORDINATE SYSTEM	22
VISUALTURN DEFAULT VIEW	22
PART GEOMETRY	23
SELECTING REGIONS.....	24
USING 3D GEOMETRY AS PART GEOMETRY	25
SETTING UP IMPORTED GEOMETRY	25
STOCK MODEL SETUP	26
SETTING UP THE MACHINE COORDINATE SYSTEM	29
CREATING MACHINING OPERATIONS.....	31
TURNING APPROACH TYPES	33
TOOLS.....	34
TOOL LIBRARY	39
FEEDS AND SPEEDS	41
CLEARANCE PLANE.....	43
ENTRY/EXIT.....	44
POST-PROCESSING	46
CAD TUTORIAL FOR CREATING A 2D PROFILE FOR TURNING.....	48
CREATING SURFACES.....	52
TUTORIAL 1: ROUGHING & FINISHING.....	55
LOADING A PART MODEL	55
CREATING TOOLS	61
CREATING THE OUTER DIAMETER ROUGHING TOOLPATH	70
SIMULATING THE OUTER DIAMETER ROUGHING TOOLPATH	77

Getting Started with VisualTurn

CREATING THE OUTER DIAMETER FINISHING TOOLPATH	79
CREATING FACE FINISH TOOLPATH	82
TUTORIAL 2: ID ROUGHING, FINISHING AND DRILLING.....	88
CREATING THE AXIAL HOLE.....	88
CREATING THE INNER DIAMETER ROUGHING TOOLPATH	96
CREATING THE INNER DIAMETER FINISHING TOOLPATH	106
TUTORIAL-3 GROOVING, THREADING AND PART OFF	110
CREATING THE OD ROUGHING TOOLPATH	110
CREATING THE OD FINISHING TOOLPATH	115
CREATING THE GROOVE ROUGHING TOOLPATH	116
CREATING THE GROOVE FINISHING TOOLPATH	119
CREATING THE THREADING TOOLPATH	121
CREATING THE PARTING-OFF TOOLPATH	124
WHERE TO GO FOR MORE HELP	126
APPENDIX I: NETWORK INSTALLATION OF VISUALTURN	127
APPENDIX II: TROUBLE SHOOTING VISUALTURN INSTALLATION	128
APPENDIX III: DESCRIPTION OF THE BROWSER TOOLBAR BUTTONS	131
SETUP TAB TOOLBAR	131
TOOLS TAB TOOLBAR.....	132
MOPS TAB TOOLBAR	132
STOCK TAB TOOLBAR.....	133
APPENDIX IV: DESCRIPTION OF OTHER TOOLBAR BUTTONS	134
THE STANDARD BAR	134
VIEW BAR.....	135
MEASUREMENT BAR.....	136
STATUS BAR	137
GEOMETRY BAR	138

Welcome to VisualTurn

Welcome to VisualTurn and thank you for choosing one of most powerful and easy to use 2 Axis turn packages on the market today.

VisualTurn is a unique, Windows-based, CAM product that seamlessly integrates toolpath generation and cutting simulation/verification, in one package that is both easy and fun to use. VisualTurn's machining technology capabilities enable you to produce toolpaths that you can send to the machine with utmost confidence. A simple and well-planned user interface makes VisualTurn suitable for use on the shop floor.

VisualTurn is a machining program targeted at the typical lathe machinist. It is ideal for machining cylindrical parts on the lathe. It can import Rhino, STL, IGES, STEP, DXF/DWG, VRML, and Raw Triangle files.

Solid models, surface models and faceted models can be imported into VisualTurn, and a wide selection of tools and toolpath strategies to can be defined when generating toolpaths. These toolpaths can then be simulated and verified, and finally post-processed to the controller of your choice.

About This Guide

This guide is designed to introduce first-time users to VisualTurn 1.0. The first part describes aspects of the user interface, machining strategies, and turning types. This is followed by several tutorials designed to familiarize you with the main features of VisualTurn.

In addition to the information provided in this guide, see the context-sensitive online help for more comprehensive explanations. You can also look at the models included in the **Tutorials** folder.

Computer Requirements

Intel Pentium compatible computer

Windows 98, NT, 2000, ME, or XP with at least 256 MB RAM.

OpenGL-compatible graphics card, displaying at least 64,000 colors

Approximately 50 MB of hard disk space.

Installing VisualTurn

To install VisualTurn software, follow these instructions:

1. Insert the CD-ROM into the CD ROM drive.
2. The setup program will automatically launch once the computer detects the CD.
3. If the program is not automatically launched, browse the CD using the Windows Explorer program and double click on the **Launch** program found in the CD. This will launch the screen shown below:



Step 1: Install Drivers (Required)

VisualTurn ships with a hardware security device called the security key (or “dongle”). This is either a 25-pin connector that connects to the parallel port of your computer, or a USB key that plugs into any USB port on your computer. You will have to install the drivers to allow VisualTurn to communicate with this security device as the first step. Click on the **Install Drivers** button on the installation screen and follow instructions to install the drivers.



USB Port Security Key



Parallel Port Security Key

Note: Plug the hardware key into your computer only after you complete installation of all software.

Once you have installed the drivers you can attach the key to your computer.

- If you have a parallel port security key and if you have any other device, such as a printer, connected through the parallel port, disconnect the device(s) and connect the VisualTurn security key to the port. Then reattach the connector of the original device(s) on top of the security key; the device(s) will continue to operate as before.
- If you have a USB port key, attach the key to any free USB port on your system

Make sure that the VisualTurn hardware key is connected to the computer. VisualTurn will not operate correctly if the security key is not connected to the computer!

Step 2: Install VisualTurn (Required)

Once you have installed the hardware key drivers and attached the key to your computer, you can install the VisualTurn product by clicking on the Install VisualTurn button on the main installation screen. Follow the instructions to complete the installation. The install program will install all the files necessary for the proper functioning of VisualTurn but also will make necessary registry modifications on your computer.

Note: Make sure you have privileges to modify the system registry before you install VisualTurn.

Step 3: Install Other Products (Optional)

Once you have installed VisualTurn you can optionally install MCU and/or Xpert DNC. These are two third party products that are included with VisualTurn.

- The MCU or Meta Cut Utilities product is a back-plot viewer that allows the user of VisualTurn to view the generated G-code graphically. This can be useful in making sure the posted output is correct before sending it to the machine tool.
- The Xpert DNC product is a single port DNC product is a communication program that allows you to send G-code files via DNC or Direct Numerical Control from your computer to the controller of the machine tool.

Step 4: Registering VisualTurn (Required)

Upon successful installation, you can run the full VisualTurn version 50 times or for 30 days without registering the product. After this period, VisualTurn will not operate anymore. VisualTurn needs to be registered with MecSoft and valid license codes obtained before it can become operable again.

To register VisualTurn, launch the product. Once VisualTurn is loaded and ready, you will see the **Enter License Codes** dialog shown below. You can alternatively access this dialog by selecting the Help option in the menu bar and choosing Register VisualTurn. The **Tries Left** field indicates the number of times you can run VisualTurn before it starts operating in demo mode.

Enter License Codes

MecSoft

Copyright 1998-2005
MecSoft Corporation

VisualTurn 1.0

License Type:

Dongle ID:

Tries Left:

Days Left:

Network Licenses Left:

Code 0:

Code 1:

Note: This registration dialog can also be invoked from the Help item in the VisualTurn menu bar.

To obtain license codes you must register the product using the Web form available at www.mecsoft.com. You can automatically launch this web form by selecting the **Request License Codes..** button in the dialog. If you have purchased the product directly from MecSoft Corporation, you will have to provide the purchase invoice number before you can be licensed. If you have purchased the

product through an authorized MecSoft reseller, please obtain the license codes from your reseller. In addition to this information make sure you also provide the Dongle ID that is shown on the registration screen.

Network Installation of VisualTurn

If you have purchased a network license of VisualTurn please follow the steps outlined in Appendix I for proper installation of the network enabled hardware key.

Troubleshooting VisualTurn Installation

If you have followed the installation steps outlined in the installation section correctly and are unable to load and run VisualTurn correctly follow the troubleshooting steps outlined in Appendix II to correct the problem.

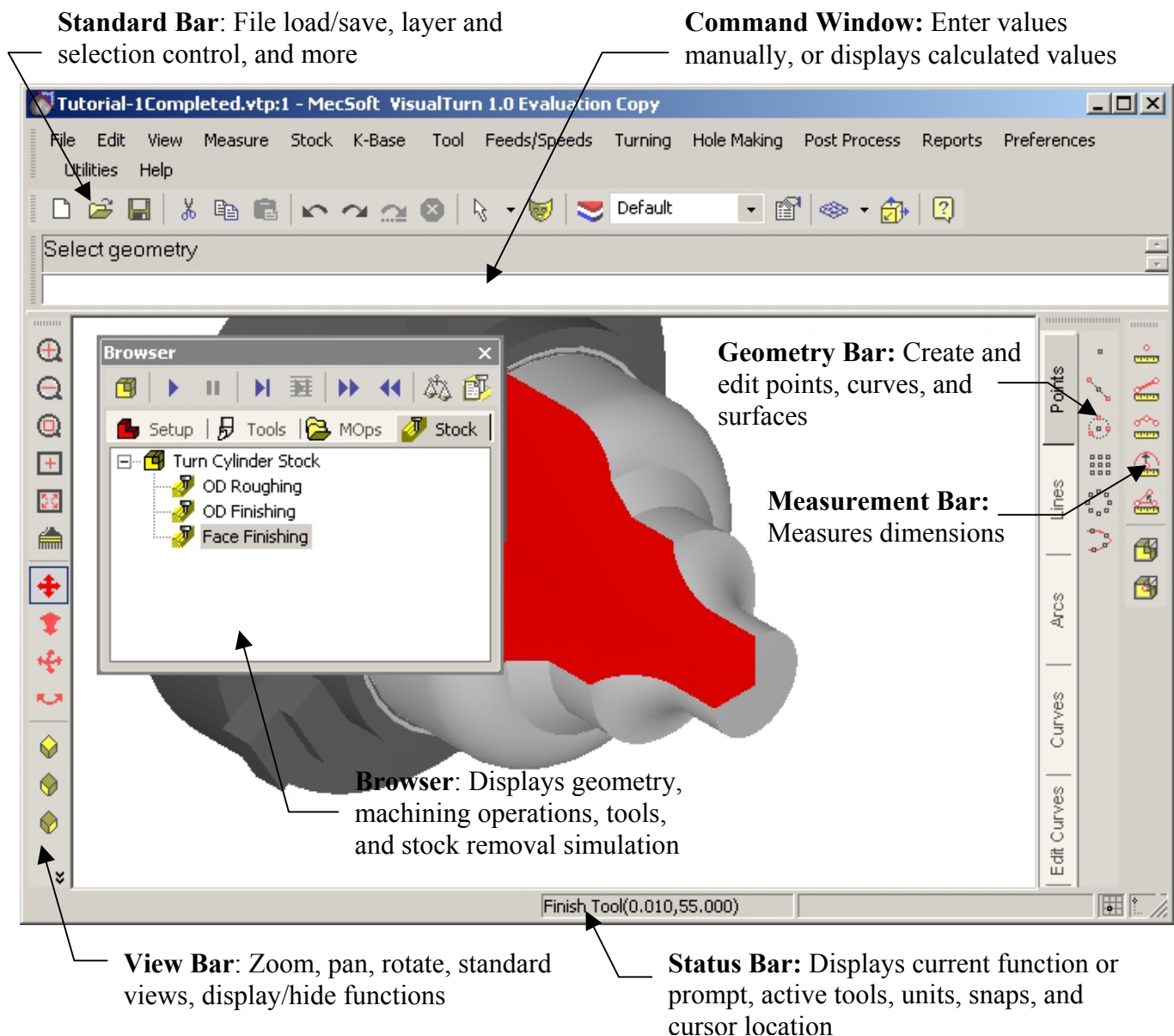
Running VisualTurn

Click on the Windows **Start** button and select **Programs**. Point to the program group containing **VisualTurn**. The name of this program group will be **VisualTurn 1.0**, unless you specified otherwise during setup. Once you locate the program group, select it and then select **VisualTurn 1.0**.

VisualTurn User Interface

VisualTurn adheres to the Windows standard for user interface design. All functions can be accessed from the menus, and common functions are accessible via toolbar icons. Most user interface settings are modal - VisualTurn “remembers” these settings and they remain active in subsequent operations unless you change them.

The main VisualTurn user interface objects are described below:

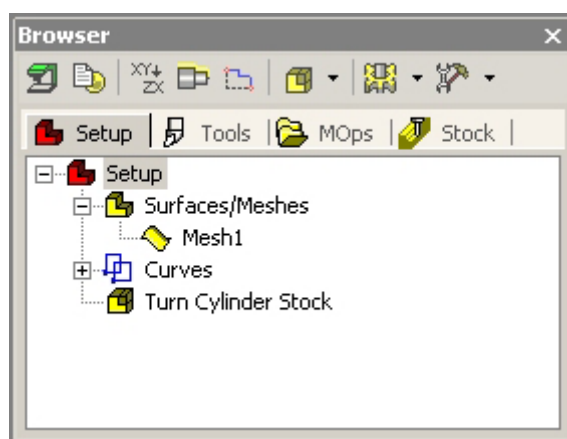


Note: You can control the display by selecting **View / Toolbars**.

VisualTurn Browser Window

The Browser is a dock-able window that allows management of various entities or objects that can be created in VisualTurn. This window is the principal window through which the user interacts with VisualTurn to program toolpaths. By default, this window will appear docked on the left hand side of the VisualTurn display when the product first comes up. This window can be undocked and move to different locations on the main screen.

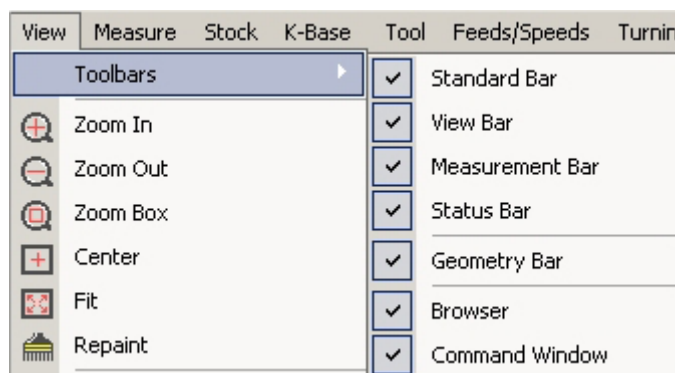
This window has four main modes of operation represented by tabs at the top of the window. These are **Setup**, **Tool**, **MOPs** and **Stock**. Selecting each of these tabs allows different views of objects in the VisualTurn database. In addition each tabbed view also incorporates a context sensitive toolbar at the top. These toolbars are groups of functions that are associated with the type of object(s) in the tab.



For an in-depth description of each of the buttons in the toolbars please refer to the on-line help of the product.

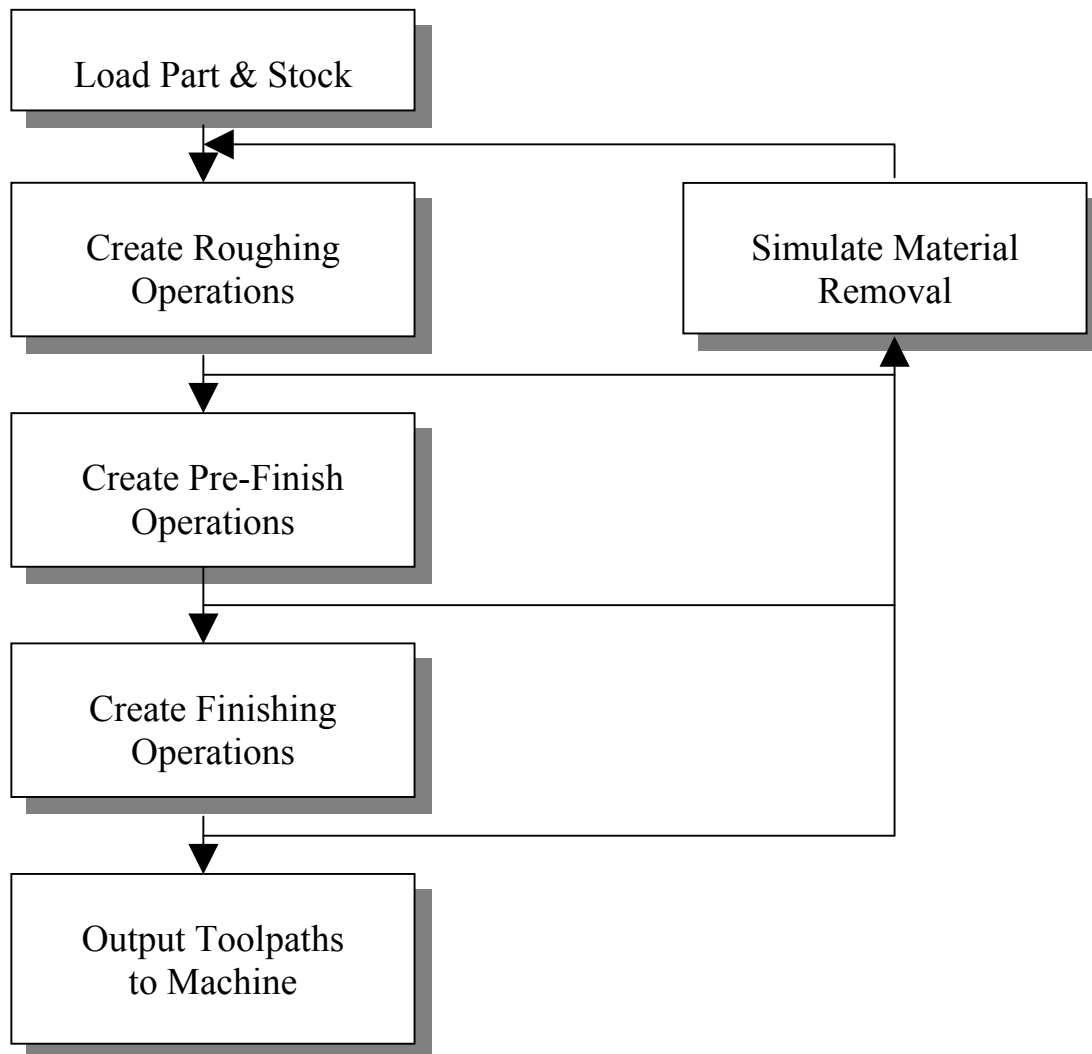
VisualTurn Toolbars

VisualTurn comes with a set of toolbars with various functions to help the programming. You can turn on/off toolbars by selecting **View -> Toolbars** in the menu bar and selecting the desired toolbar. A description of each of these toolbars and their buttons is described in the Appendix of this document.



VisualTurn Workflow

The manufacturing process aims to successively reduce material from the stock model until it reaches the final shape of the designed part. To accomplish this, the typical machining strategy is to first use large tools to perform bulk removal from the stock (roughing operations), and then use progressively smaller tools to remove smaller amounts of material (pre-finish operations). When the part has a uniform amount of stock remaining, a small tool is used to remove this uniform stock layer (finish operations).



This machining strategy is what you program using VisualTurn. You can also simulate material removal to visualize how the stock model will look at any time during the process. This provides valuable feedback that can help you choose the most appropriate machining strategy.

Typical Scenario

Rough machining can be done by **Roughing** operations, using a turning tool with a relatively large nose radius. These rough operations can be followed by subsequent roughing operations, either using the same tool or a smaller tool.

Final finishing of the part can then be performed by using one or more **Finishing** operations. Finishing operations typically use tools with smaller nose radius so as to obtain a better surface finish and tighter tolerance levels.

Depending on the geometry of the part and/or machining operations desired, **Groove Roughing**, **Groove Finishing**, **Follow Curve**, **Threading** and the **Hole-Making** operations can be considered. After completing all the machining operations, the final part is cut off from the rest of the bar stock by using the **Part-Off** operation.

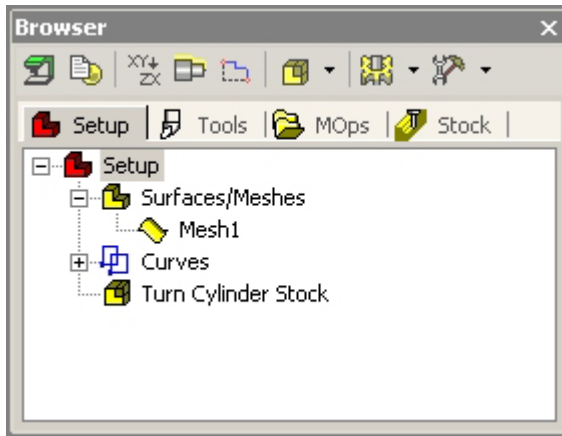
Once all of the operations are completed, you can go back and review the operation sequence, re-order and/or change operations if desired, simulate the material removal, and post-process the toolpaths. The Browser can be used to manage these operations.

Programming Workflow

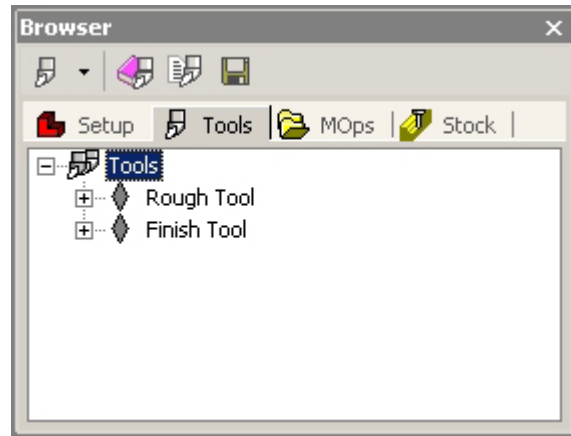
Once the part is loaded, the typical workflow is reflected in the layout of the tabs and toolbars of the Browser window. The workflow is designed to allow the user to work starting from the left most tab and ending at the right most tab. Additionally each of the functions in each of the toolbars corresponding to each tab is also best accessed in order from left to right.

Thus the user typically would start with the **Setup** tab and access each of the buttons, optionally, in the toolbar that appears when this tab is selected in sequence from left to right. Once the setup functions are completed, the user will then proceed to the **Tools** tab to create, select and save tools to be used in the machining. After this the user will proceed to the **MOps** or **Machining Operations** tab and commence programming the part. Once a program is completed the user can switch to the **Stock** tab to perform the material removal simulation and/or the tool animation to preview the toolpath before sending it to the machine tool.

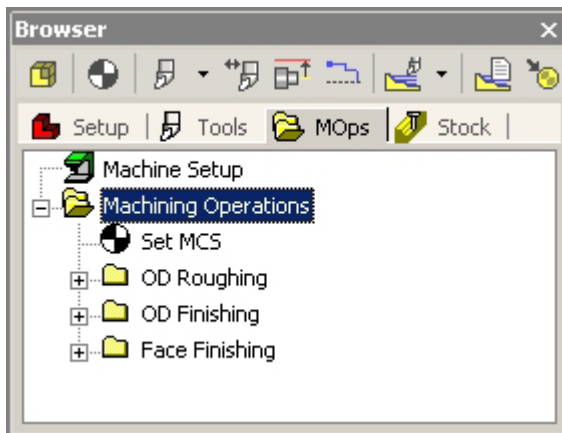
Step 1: Setup before programming



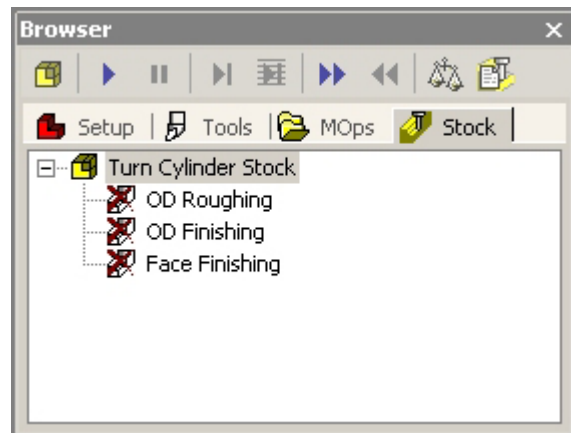
Step 2: Create, select and save tools



Step 3: Create machining operations



Step 4: Simulate machining operations



Post-Processing

Once the machining operations have been created and verified, they can be post processed to create G-code files. These G-code files can then be sent to the controller of the machine tool to drive the actual machine tool.

Machining Methods

There are two major classes of machining operations that can be created in VisualTurn – Turning and Hole-Making. Turning operations are used to remove material from cylindrical shaped stock on a lathe machine to get the desired shapes. Hole-Making operations are used to create axial hole features in the part.

Turning Operations

Turning operations are operations used to create the shape of the part. All 2-axis turned shapes can be represented as a surface or solid of revolution. Turning operations are used to create the shape out of an initial cylindrical stock model.

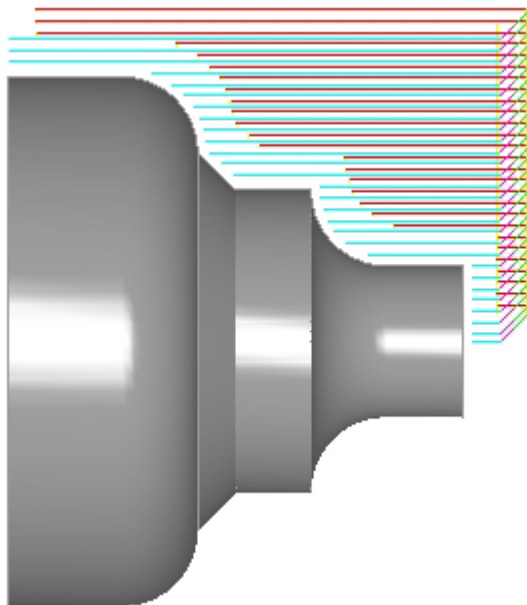
The various types of operations available in VisualTurn are described below:



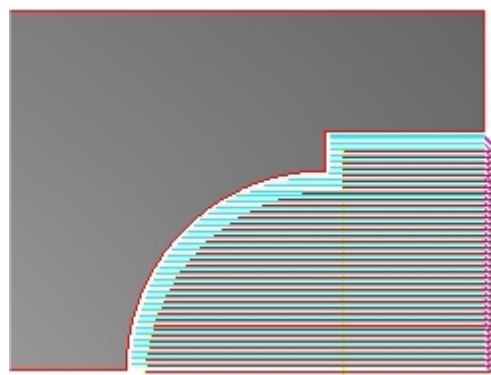
Roughing

This operation is typically performed to remove material from the stock, thus is characterized by larger depth of cuts. Typically material is roughed out in multiple cuts. This type of machining is very efficient for removing large volumes of material, and is typically performed with a large radius tool. Roughing is typically followed by finishing toolpaths.

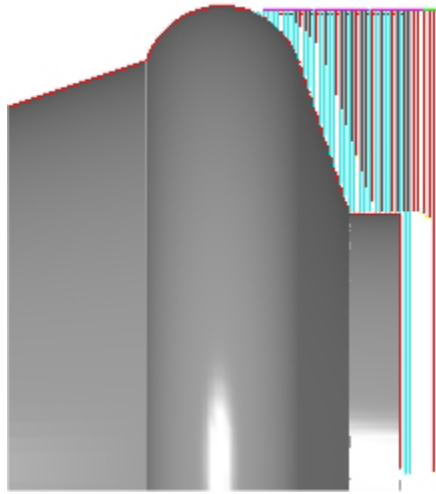
Both part and stock geometry are used to determine the regions that can be safely machined. Roughing can be of 3 types: **OD Roughing**, **ID Roughing**, and **Front Facing** (Face Roughing)



Outer Diameter (OD) Roughing

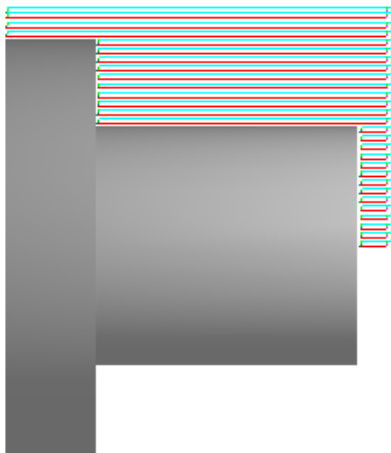


Inner Diameter (ID) Roughing

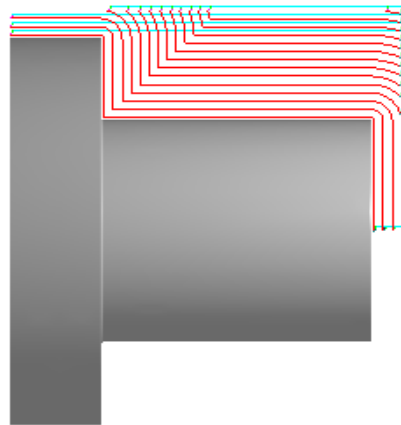


Face Roughing

Cut patterns: Two types of cutting patterns are available: **Linear** (parallel to the Z-axis), **Offset** (parallel to the part region).



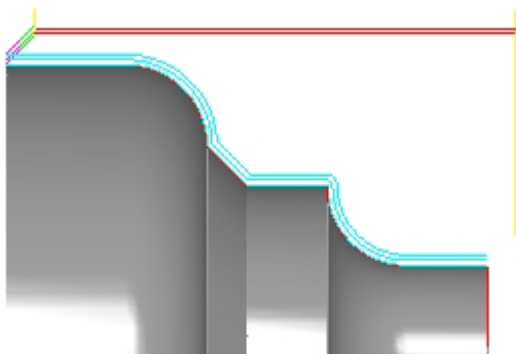
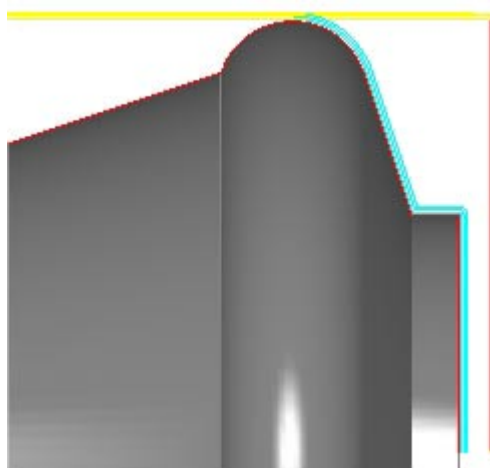
Roughing – Linear



Roughing – Offset

 Finishing

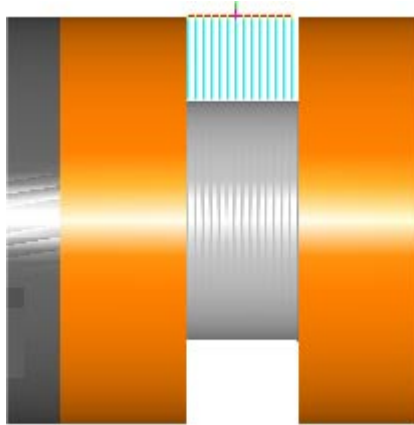
This operation is performed after roughing operation. Only the part geometry is taken into consideration in this machining operation and is offset to calculate the finishing tool-path. This operation is characterized by smaller depth of cuts to obtain tighter tolerances and better surface finish.

**OD Finishing****ID Finishing****Face Finishing**



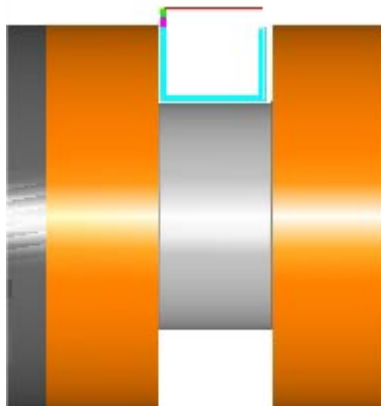
Groove Roughing

This operation is performed to machine grooves on the part. The grooves are typically used to slide/fit one part into another to obtain the required assembly.



Groove Finishing

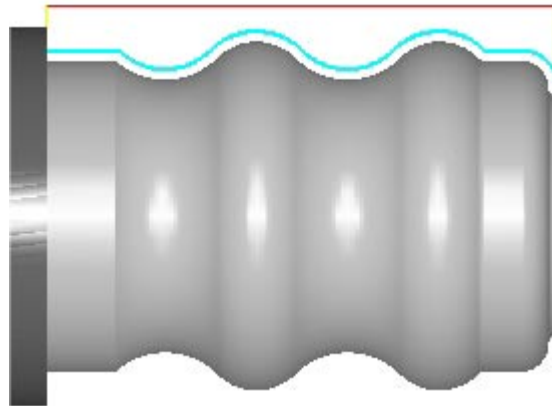
This operation is used to finish the grooves. This operation is performed after the Groove Roughing operation.





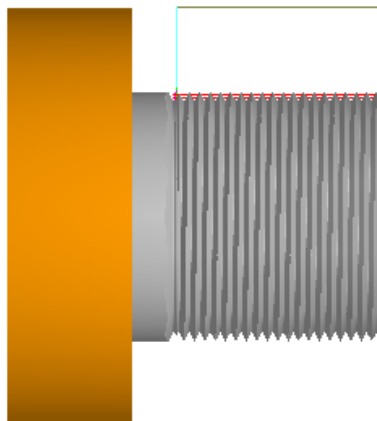
Follow Curve

This operation is performed in difficult to reach areas. The tool is driven about the curve with no offsets applied to the curve.



Threading

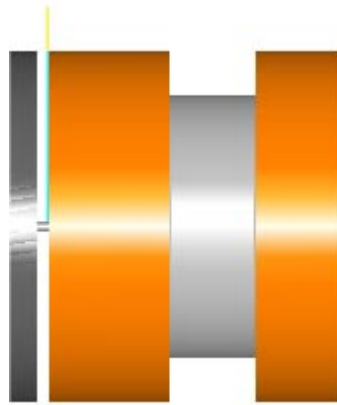
This operation is performed to machine threads on the part. Threads are used as fasteners for assembly purposes.





Part Off

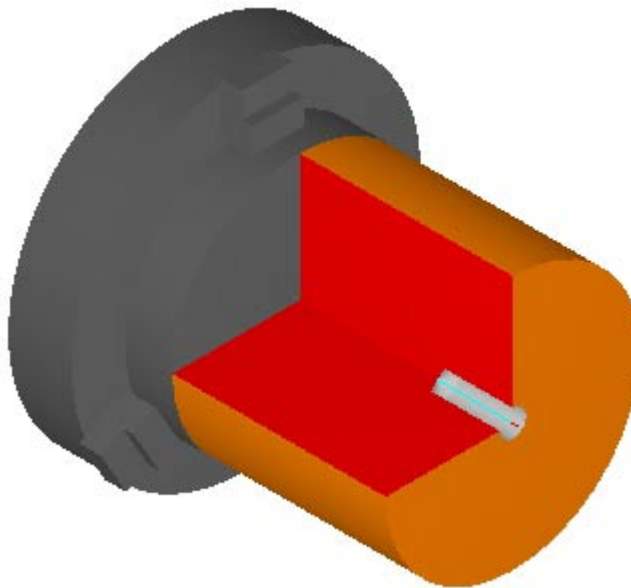
This operation is performed to cut off the finished part from the rest of the bar stock.



All the turning operations as mentioned above, except **Part Off**, can be carried on the **Outer Diameter**, **Inner Diameter** or the **Front Face** of the work-piece.

Hole-Making Operations

Hole making operations in a 2-axis turning machine are always performed axially. That is only holes that are aligned with the rotation axis of the part and also on the front face of the part can be created. An example of an axial hole is shown below. The part is chucked on the lathe as usual and the hole-making tool is moved along the axis of rotation to create the hold.



The various types of hole-making operations available in VisualTurn are described below:



Drilling - The following drill cycles are available:

Standard: Used for holes whose depth is less than three times the tool diameter.

Deep: Used for holes whose depth is greater than three times the tool diameter, especially when chips are difficult to remove. The tool retracts completely to clean out all chips.

Counter Sink: Cuts an angular opening at the end of the hole.

Break Chip: Similar to **Deep** drilling, but the tool retracts by a set clearance distance.



Tapping

A Tap cycle is used to drill threaded holes in the part, clockwise or counter-clockwise.



Boring

A Bore cycle is used to form shapes inside a hole. The following boring cycles are available:

Drag: The tool is fed to the specified depth at the controlled feed rate. Then the spindle is stopped and the tool retracts rapidly.

No Drag: The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and then retracted.

Manual: The tool traverses to the programmed point and is fed to the specified depth at the controlled feed rate. Then the tool stops and is retracted manually.



Reverse Boring

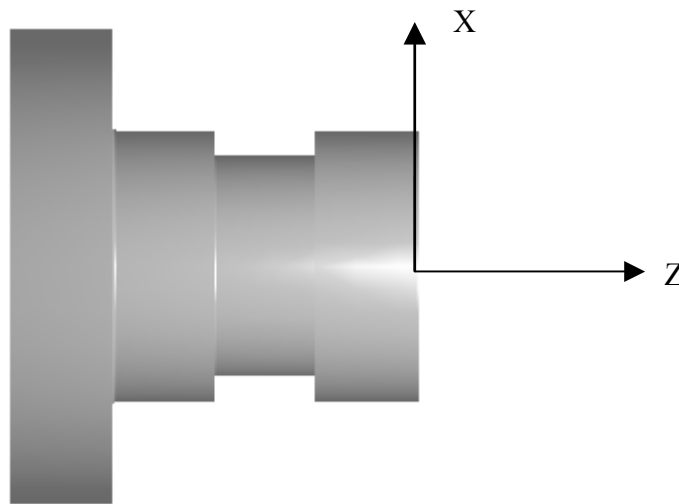
This is simply a Bore cycle in the reverse direction. The spindle is oriented to the specified angle and moves rapidly to the feed depth and moved to the part. The spindle is turned on and the cycle is started.

Key Concepts in VisualTurn Programming

Before attempting to use VisualTurn there are a few key concepts that are used in VisualTurn that need to be understood. Some of these concepts will be familiar to lathe programmers and are explained here because they are essential for the proper use of VisualTurn.

Turning Coordinate System

CNC turning centers use the Cartesian coordinate system for programmed coordinates but they are typically different from that used in milling. Turning centers follow the convention that axis of rotation that is aligned with the spindle is designated as the Z axis. Secondly the axis perpendicular to this axis along which the tool travels to cut into the stock is designated the X axis. Thus the part is rotated about the Z-axis of the lathe machine. Moving the tool along the Z-axis provides the direction of feed and moving it along the X-axis provides the depth of cut. This is shown below.



VisualTurn Default View

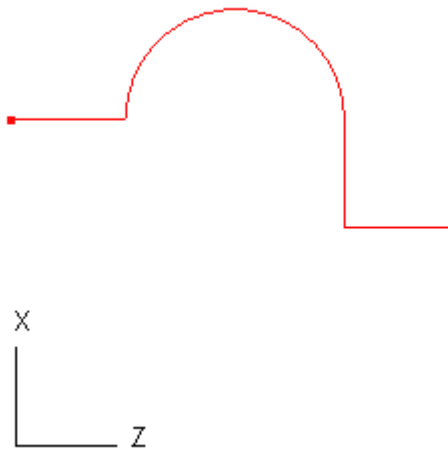
VisualTurn uses the Top view as the default view. This top view is additionally setup to be aligned with the turning coordinate system. That is the origin of the screen is located at the center of the screen and the Z axis goes from left to right and the X axis goes from bottom to top. This display setup is not typical in design systems where the Top view is aligned with the XY axes of the world coordinate system. This view setup is used in VisualTurn to allow the turning center programmer to work in turning center coordinates rather than in the XY coordinates of the design system.

It should be noted that this convention might sometimes be disorienting for users who are used to visualizing their design parts in the normal XY aligned display rather than the ZX aligned display.

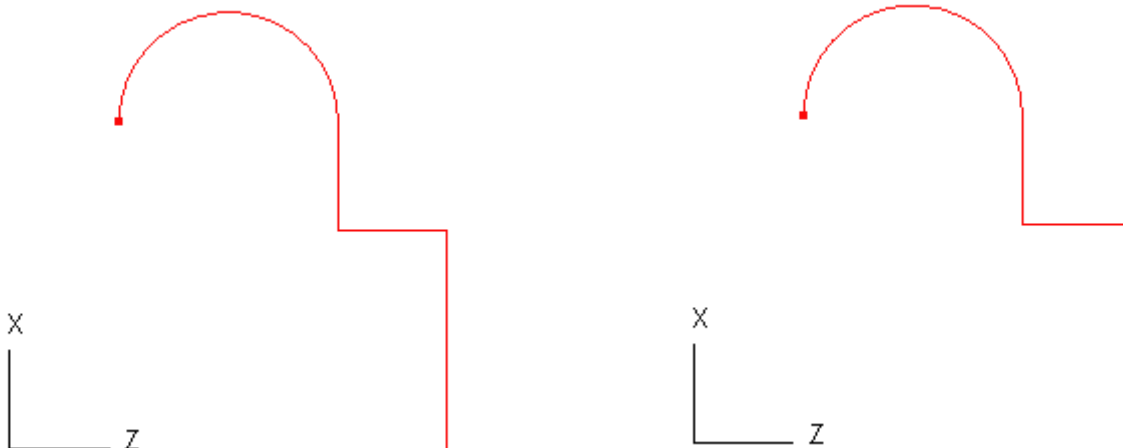
Note: VisualTurn's Top view is by default aligned with the ZX turning center coordinate system.

Part Geometry

VisualTurn requires regions/curves that define the part geometry. Since all parts that can be created in a 2-Axis turning machine are solids of revolutions, it is enough to describe the profile that needs to be revolved to create this shape. The profile can be created in VisualTurn as a region or curve. Furthermore VisualTurn places a further restriction that these part regions need to be constrained to lie **only** in the first quadrant of the ZX plane. That is, one end point of the region must touch the X axis and the other end of the profile should touch the Z axis. VisualTurn will be unable to process a part region that does not follow these restrictions.



Valid part region: Region correctly positioned in the first ZX quadrant touching both the X and Z axis



Invalid part regions: Region is not touching the X axis and/or Z axis

Note: Part regions need to be constrained to the first quadrant of the ZX coordinate system.

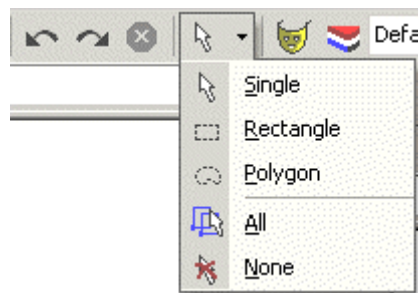
Getting Started with VisualTurn

Part Regions can be imported or can be created within VisualTurn using the Geometry creation and editing tools of VisualTurn. You can select a part of the region or the whole region for machining purposes. The **Geometry Bar** contains all the tools you need to create regions, in addition to other types of geometry. It is located to the right of the graphics area. If you do not see this toolbar, select **View / Toolbars / Geometry Bar**.

Selecting Regions

Regions must be selected in order for them to be used in an operation. Creating a region does not make it active; you must use one of the **Select Regions** tools before creating the toolpath.

Region selection tools can be accessed from the **Select Regions** icon. These tools are also available in the **Mops** tab.



When selected, a region is highlighted in yellow (depending on the color preferences set). Note that any selected regions remain active until deselected, so when you want to activate different regions be sure to deselect any you do not want.

The following region tools are available:



Single: You can select existing regions by picking them manually. Multiple regions can be selected by pressing Ctrl.



Rectangle: Selects all regions within a defined rectangle.



Polygon: Selects all regions within a defined polygon.



All: Selects all regions defined in the model.



None: Deselects any selected regions.

Using 3D Geometry as Part Geometry

VisualTurn has the capability of extracting 2D profile from a 3D geometry. Since VisualTurn uses wire frame geometry (regions) to define the part geometry, created or imported 3-D geometry cannot be used directly. Using the **Slice** and **Resolve Part Region** tools provided in the **Setup** tab toolbar of the browser window the user can easily create 2D geometry that can then be used as input to the VisualTurn toolpath generation methods. The **Slice** button slices the input 3D model with an infinite XZ plane and creates one or more regions/curves in VisualTurn. The **Resolve Part Region** button extracts a part region that is completely inside the first quadrant of the ZX coordinate system by either trimming the region against the axes and or extending the ends of the regions to the closest axes. Once resolved these part regions can then be used as part geometry for the VisualTurn toolpath generation methods.

***Note:** Refer to Tutorial 1 for more information on **Slice** and **Resolve Part Region** commands*

Setting up Imported Geometry

As mentioned earlier design systems use the normal Cartesian system for designing parts. So parts models will usually have the axis of rotation aligned with the global X axis and the radial direction aligned with the global Y axis. When you import such a model into VisualTurn there is an easy way of converting the geometry such that the axes are properly aligned with the turning center coordinate system. Selecting the **Convert XY to ZX**  button in the **Setup** tab toolbar of the Browser window will perform this transformation automatically. This command works for both 2D & 3D geometry. Follow these steps for performing the necessary transformation.

Steps for Converting XY to ZX for a 3D geometry

1. Load the 3D geometry into VisualTurn.
2. Select  Convert XY to ZX from the Setup Tab of the VisualTurn Browser.

This will orient the curve to the ZX (lathe coordinate system)

Steps for Converting XY to ZX for a 2D geometry

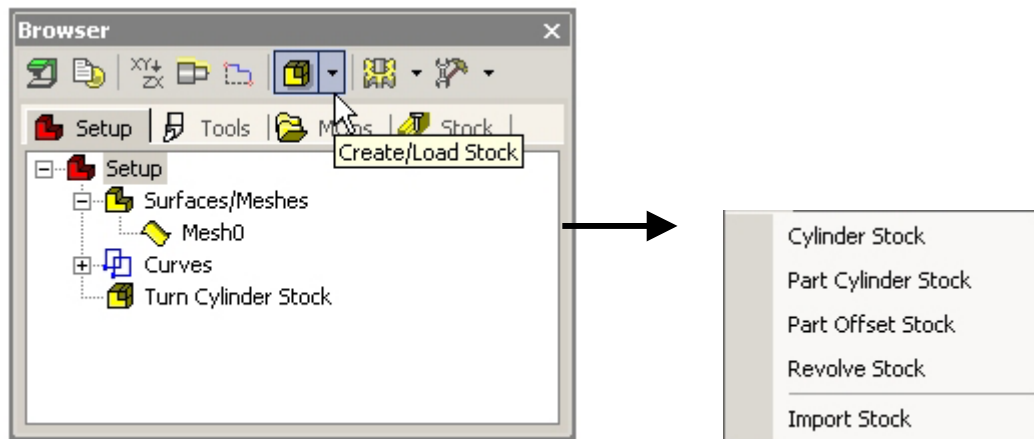
1. Load the 2D geometry into VisualTurn.
2. Select  Convert XY to ZX from the Setup Tab of the VisualTurn Browser.
3. Make sure the 2D trace touches the X and Z of the coordinate axis.
4. Use the transformation tools from the Edit menu to move the geometry to the 1st quadrant of the lathe coordinate system if necessary.

Stock Model Setup

“Stock” represents the raw stock from which the part will be manufactured. Stock geometry can either be created within VisualTurn or imported from an external file. Stock can also be created within VisualTurn by entering the length and radius of the cylindrical stock or as the bounding cylinder of the part. You can also define stock as an offset, both in the radial and axial direction of the part geometry, to simulate casting or forging raw stock model.

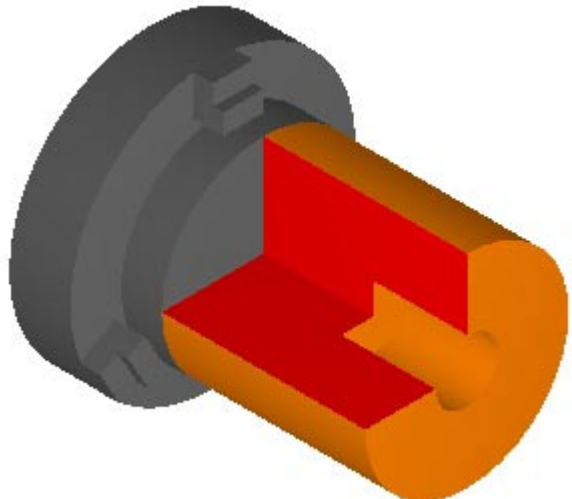
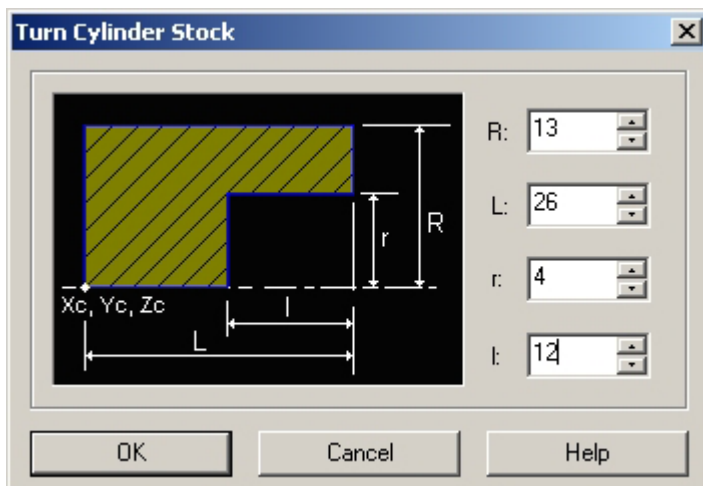
Note: Stock can be created only after part geometry is created or loaded in **VisualTurn**. You must define a stock model before creating **Turn Roughing** and **Groove Roughing** operations. All other operations can be created without first creating a stock model.

To create stock models select **Create/Load Stock** from the **Setup** tab to select the stock type. (This tool is also available on the **Stock** tab of the Browser.)

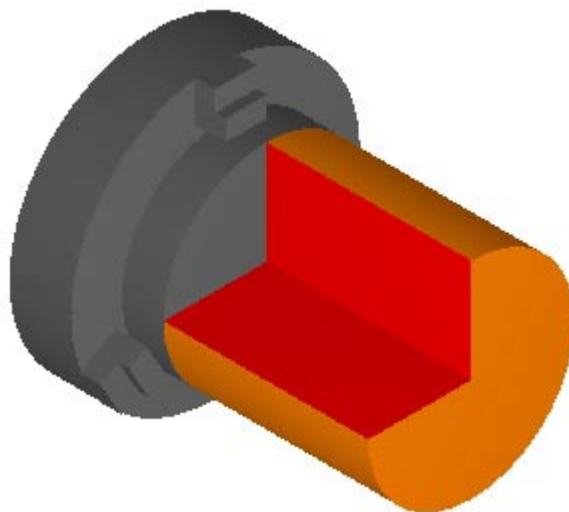
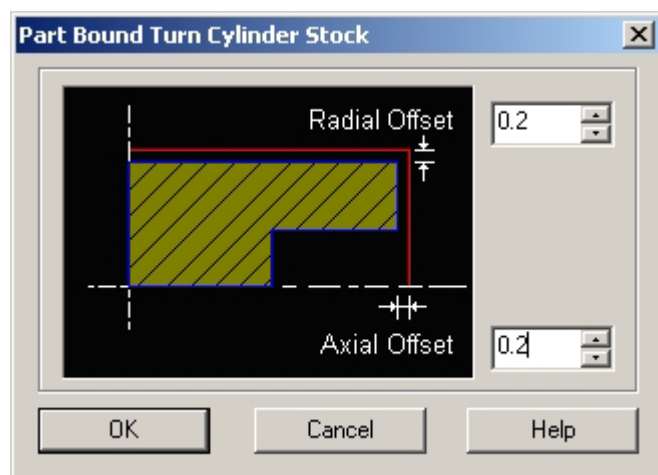


The various types of stock models that can be created in VisualTurn are described below:

Cylinder Stock: In this type of Stock model user can specify the **Radius** (Outer and Inner) and **Length** (Major and Minor) for the stock.



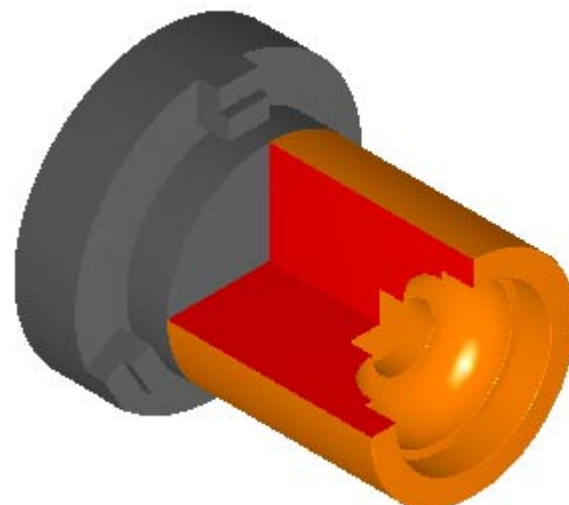
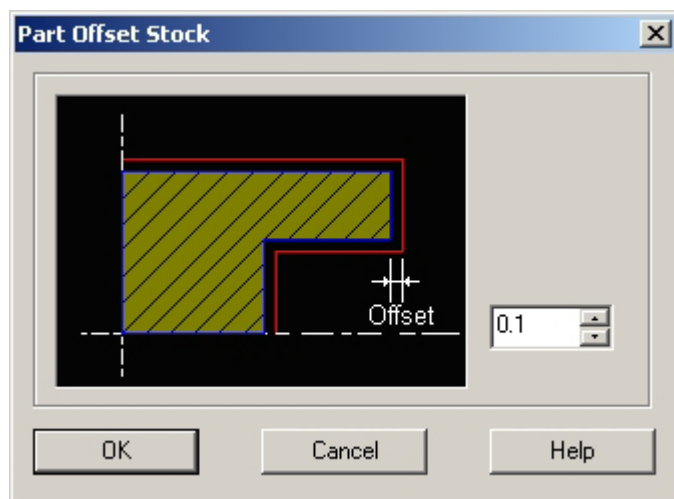
Part Cylinder Stock: Here a cylinder that encompasses the part completely in the Z and the X axis can be created. The user can additionally specify a **Radial Offset** and/or **Axial Offset** for the stock.



Part Offset Stock: User needs to select a 2D part region before creating a Part Offset Stock. User can then specify offset value to create the stock model. The part region will be revolved around the Z axis after an uniform offset applied to the region

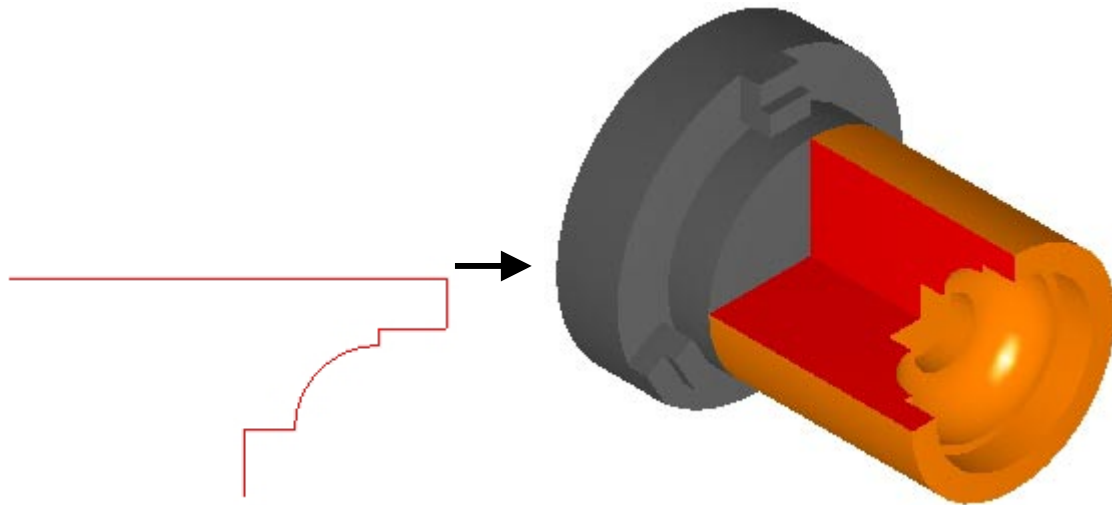


2D Part Region



Getting Started with VisualTurn

Revolve Stock: User needs to select a 2D profile before creating a Revolve stock. The above curve is used to create a revolve stock for this example.



2D Profile

Import Stock: User can import STL solid models (ASCII and binary) for stock geometry. Surfaces can be imported from IGES or Rhino 3DM. Faceted (triangulated) models can be imported from VRML, Raw Triangle, DXF / DWG facet data, or Rhino Mesh. Note that in-order for the import stock to work correctly it needs to be a water tight model. Gaps between faces of the model will result in problems during the creation of the stock model.

***Tip:** Stock is used for simulation, and its display involves data-intensive rendering. This can slow down VisualTurn's performance. Therefore, we recommend turning off the stock display when not needed.*

Stock Model Display: The stock model is created and switches to the **Stock** tab of the Browser. The stock is displayed as a cylinder positioned at the reference point of the lathe machine. Its color can be set in the **Color Preferences**. If you are unable to see the stock, make sure the **Hide Stock** toggle icon in the **View** bar is turned off.



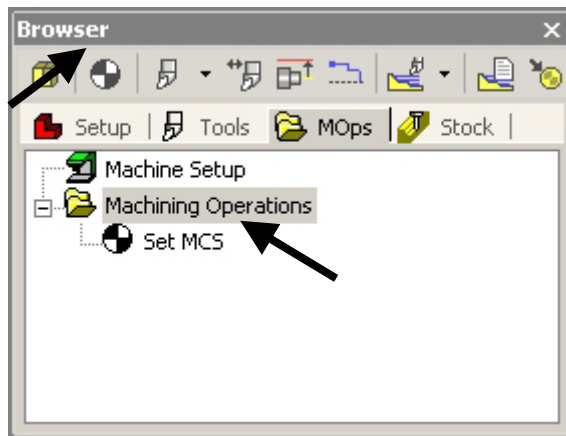
Setting up the Machine Coordinate System

Before we start machining, the machine co-ordinate system has to be set. This allows us to define the program zero, with respect to which the tool-paths are calculated and output. The program zero is variously called work datum, program reference point and work zero etc. This point defines the coordinate origin of the program. All program points output to the machine tool are described with respect to this point.

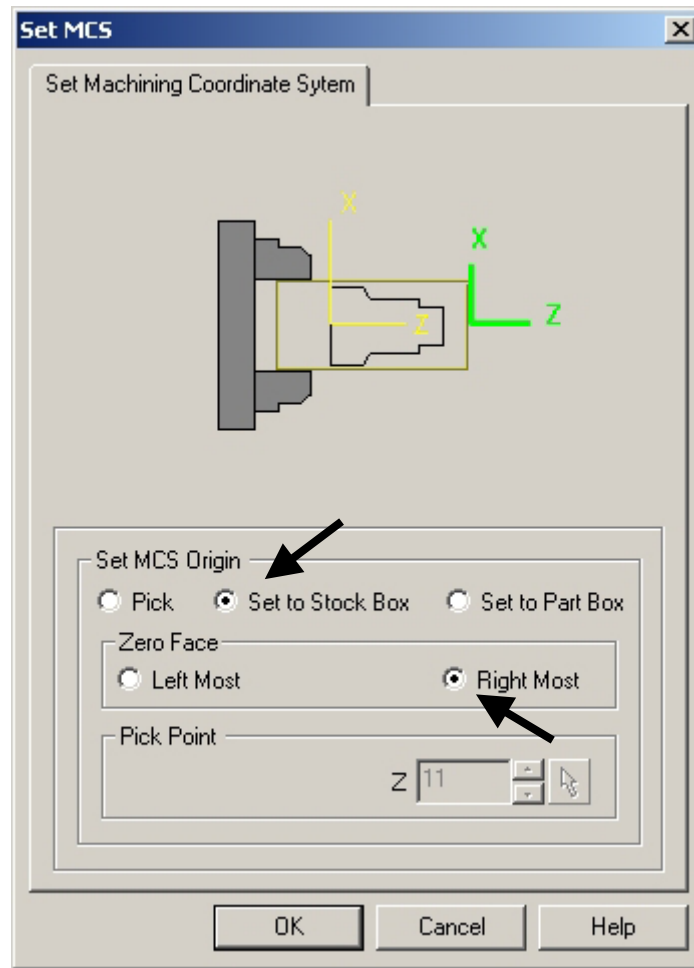
In typical shop floor practice, this program zero point is set at a position such that the X coordinate of this point is on the axis of rotation and the Z coordinate of this point is flush with the right most face of the work-piece or stock.

VisualTurn allows the user to specify this program zero point conveniently by using a dialog. To set the program zero, or to set the MCS follow these steps:

1. Click on the **Set MCS** icon in the **Mops** tab of the Browser Bar or Double-Click on the Set MCS under Machining Operations



2. This gives the user different options to set the machine zero. The user can set the zero to the left/right face of the part/stock box or pick the point directly.
3. As mentioned above the general shop floor practice is to set the **MCS** origin to the **Stock Box** and Zero Face to **Right Most** face of the part. Click **OK**.



This should align the MCS with the rotation axis and the right most face of the stock model.

Note: Tool X and Z offsets that are required for each tool in the tool turret of a CNC turning center have to be measured from this point. These tool offsets are necessary to be programmed in the controller correctly for proper cutting of the part when using an automatic tool turret in a CNC turning center.

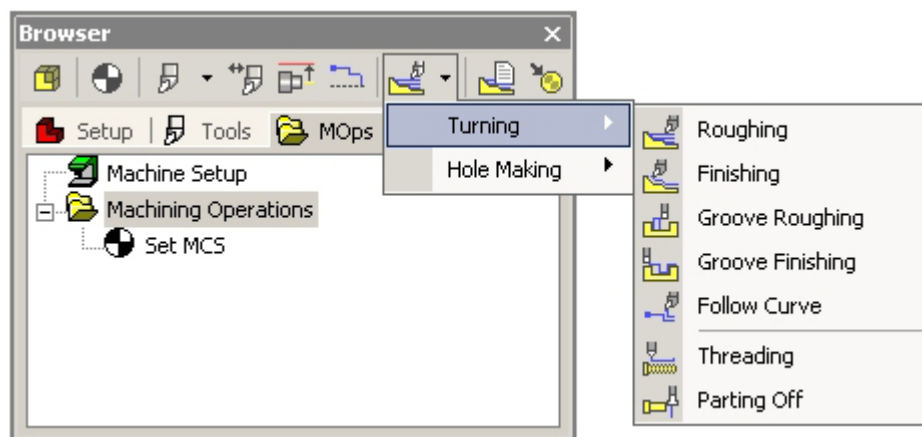
Creating Machining Operations

VisualTurn allows users to create machining operations for turning and hole making operations. The user needs to make sure that all setups described previously have been completed before proceeding to creation of machining operations. Additionally the user first needs to select the following items before proceeding with the program creation:

1. Stock model if programming a roughing operation*
2. Correct tool for the operation
3. Choose the correct feeds and speeds setting
4. Choose the clearance plane specification
5. Part Region that defines the part to be machined.

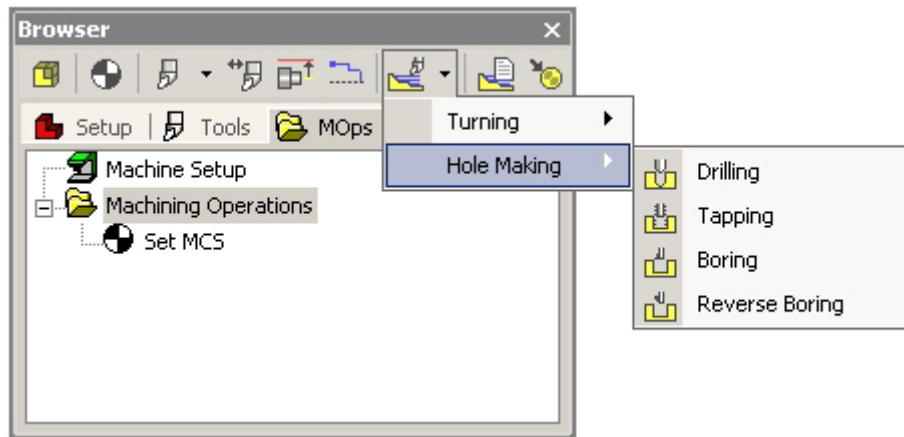
** A stock model is a pre-requisite only for creating **Turn Roughing** and **Groove Roughing** operations. All other operations can be created without first creating a stock model.*

Once all of these items have been created or made active machining operations can be created. All turning operations can be accessed using the Machining Operations toolbar button in the toolbar belonging to the Mops tab of the browser as shown below.



Getting Started with VisualTurn

All hole-making operations can be accessed using the Machining Operations toolbar button in the toolbar belonging to the Mops tab of the browser as shown below.

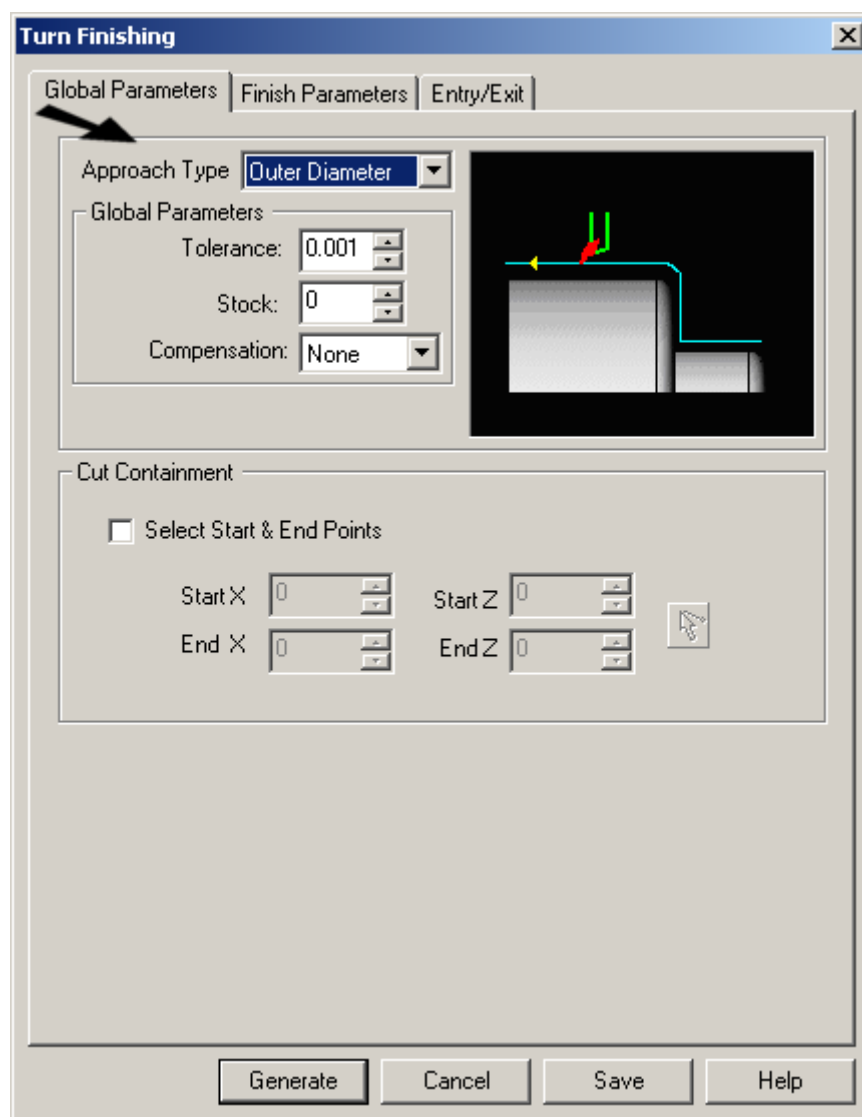


Note: Refer to previous chapter for a detailed description of each of the machining types

A description of each of the objects needed prior to creating machining operations is detailed in the following sections of this chapter.

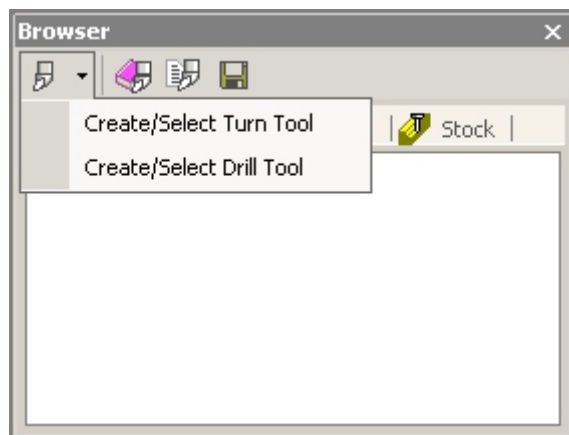
Turning Approach Types

The approach type of an operation defines the axis (X or Z) about which the tool will approach the part for machining. There are 3 types of approaches that are typically used. These are Outer Diameter (OD), Inner Diameter (ID) and Front Facing (Face). In the OD and the ID approach types the tool will approach and retract along the X axis. In the case of OD the approach will be along the positive X axis while in the case of ID it will be along the negative X axis. In Face approach the tool will approach and retract along the negative Z axis. The approach type is a necessary parameter and will have to be defined in every turn operation in VisualTurn. An example of setting the approach type in the VisualTurn turn finishing dialog is shown below:

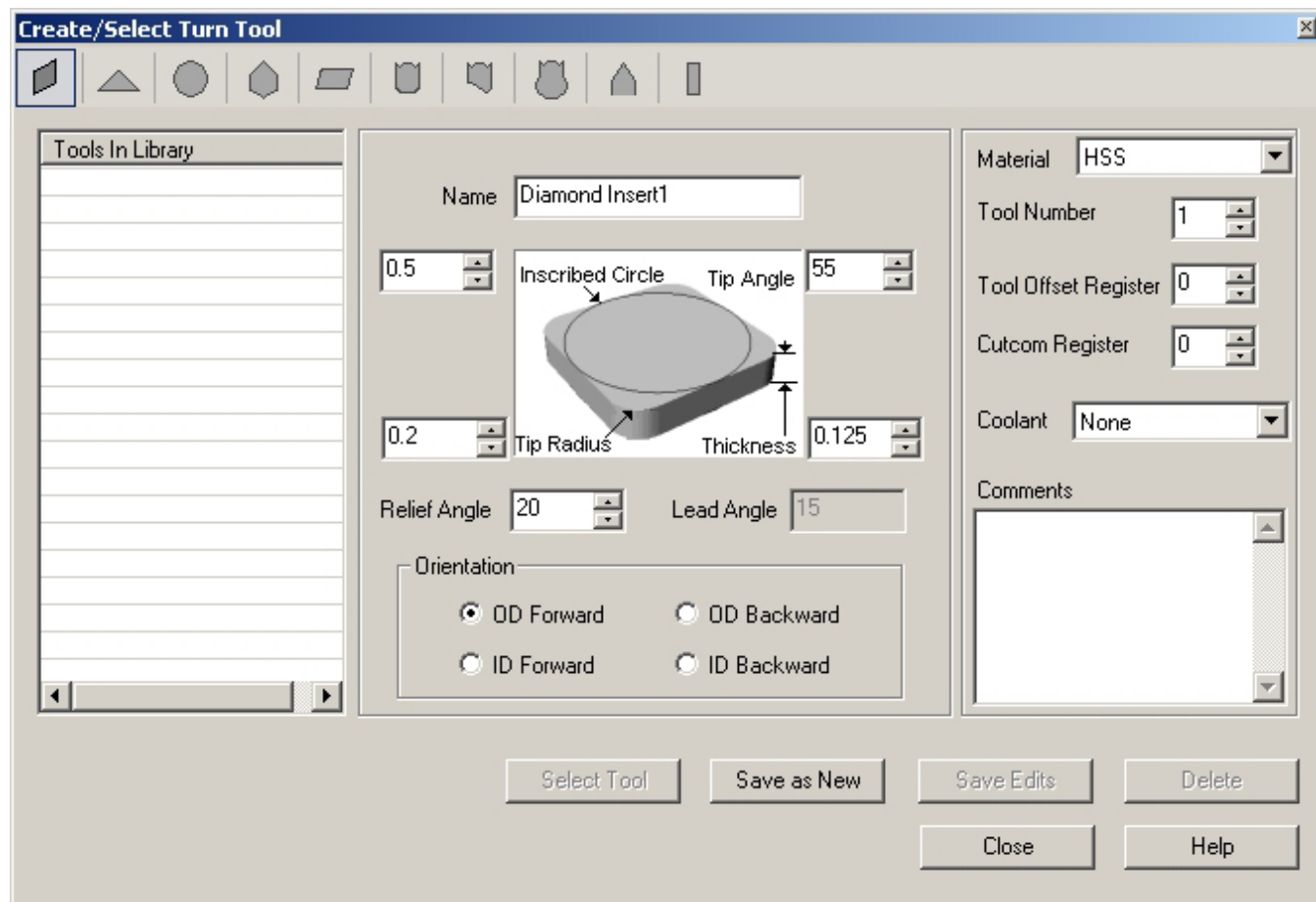


Tools

VisualTurn supports numerous types of turning and drilling. To access the tools creation command, switch to the Tools tab in the browser window and select the first button in the toolbar.



Selecting the Turn tool brings up the dialog shown below. Use the toolbar at the top of the tools dialog to select the desired tool type.



Various turn tool types such as Turning inserts, grooving, threading and parting off tools can be created. The supported types are:

Name:

Inscribed Circle Tip Angle

Tip Radius Thickness

Relief Angle: Lead Angle:

Orientation

☒ OD Forward ☐ OD Backward

☐ ID Forward ☐ ID Backward

Diamond Insert

Name:

Nose Angle Inscribed Circle

Tip Radius Thickness

Relief Angle: Lead Angle:

Orientation

☒ OD Forward ☐ OD Backward

☐ ID Forward ☐ ID Backward

Triangular Insert

Name:

Radius Thickness

Circular Insert

Name:

Nose Angle Inscribed Circle

Tip Radius Thickness

Relief Angle: Lead Angle:

Orientation

☒ OD Forward ☐ OD Backward

☐ ID Forward ☐ ID Backward

Trigon Insert

Getting Started with VisualTurn

Name:

Length: Width:

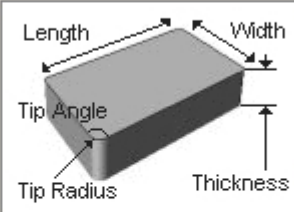
Tip Angle: Tip Radius: Thickness:

Relief Angle: Lead Angle:

Orientation:

☒ OD Forward ☐ OD Backward

☐ ID Forward ☐ ID Backward



Parallelogram Insert

Name:

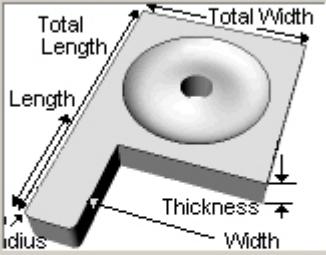
Total Length: Total Width:

Length: Thickness: Width:

Radius:

Program Point:

☒ Left ☐ Center ☐ Right



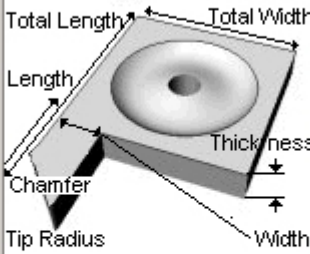
Groove Insert

Name:

Total Length: Total Width:

Length: Thickness: Width:

Chamfer: Tip Radius:



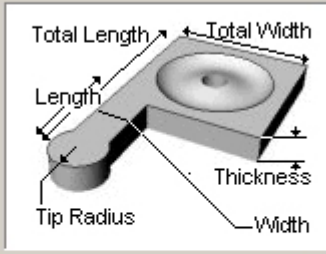
Groove Chamfer Insert

Name:

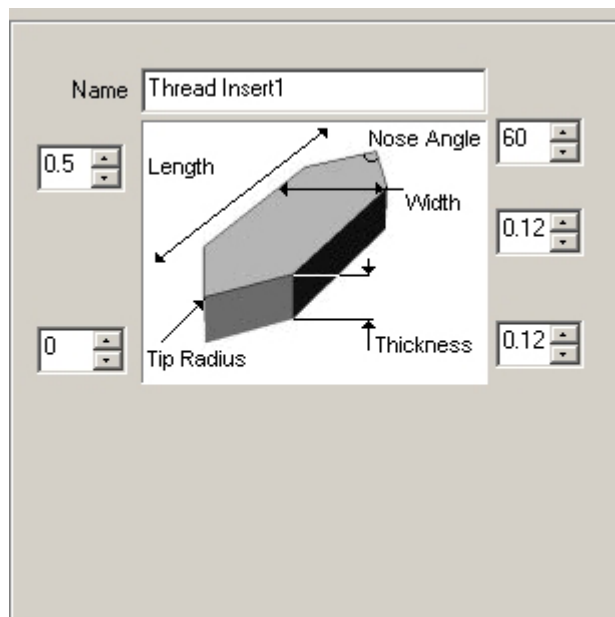
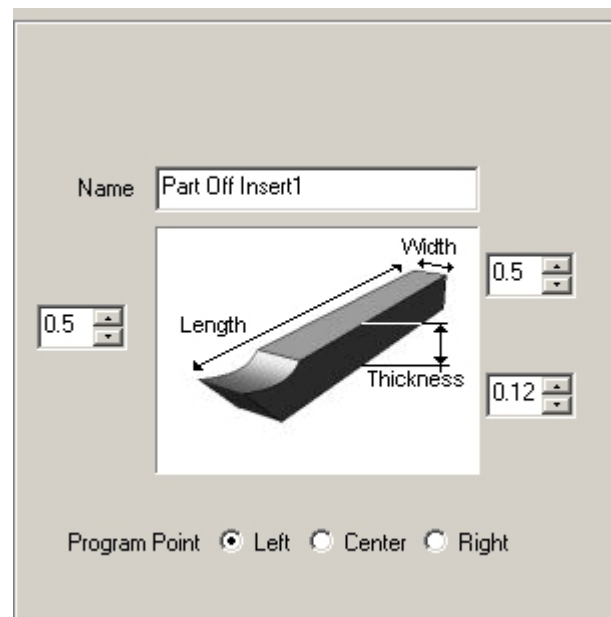
Total Length: Total Width:

Length: Thickness: Width:

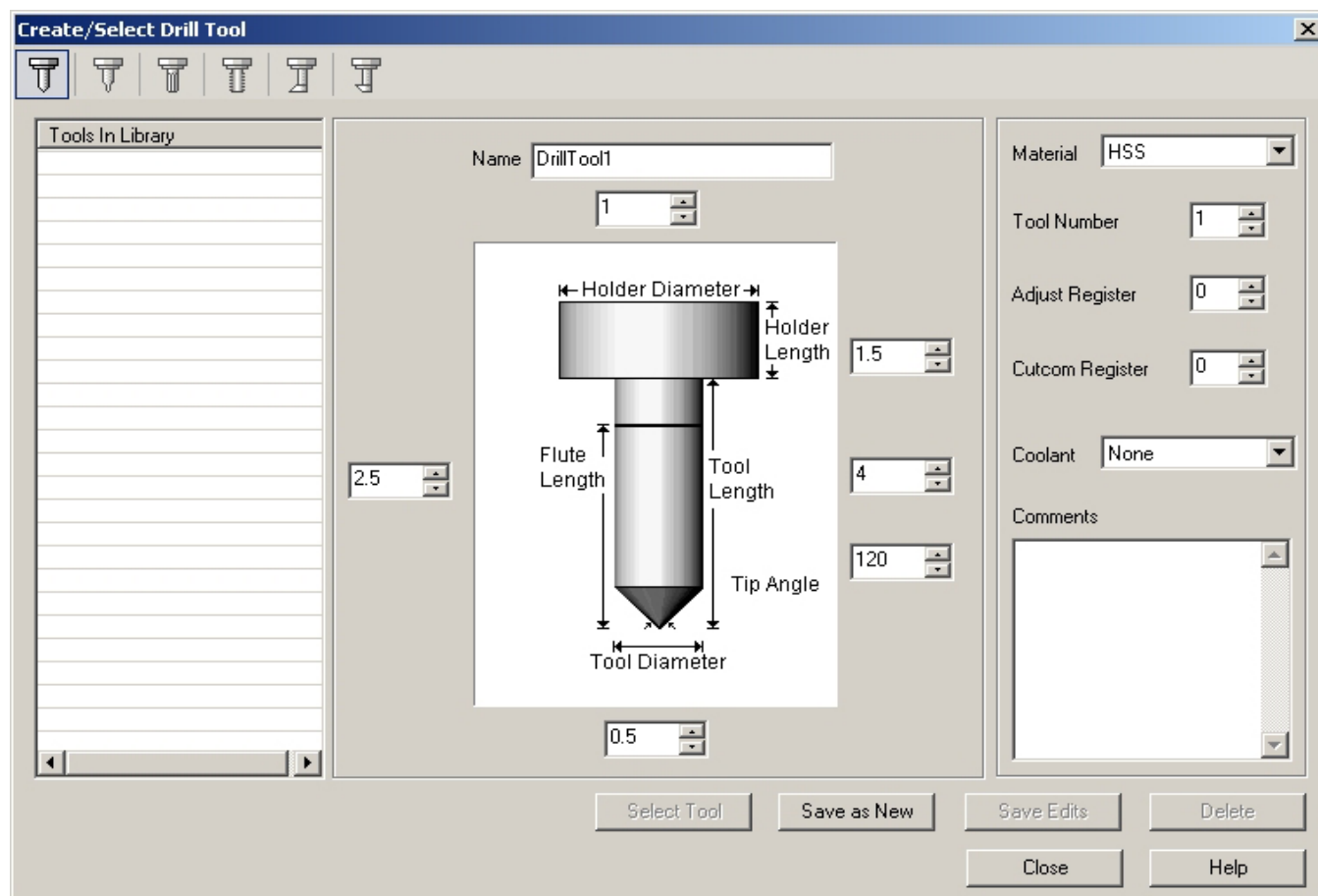
Tip Radius:



Groove Round insert

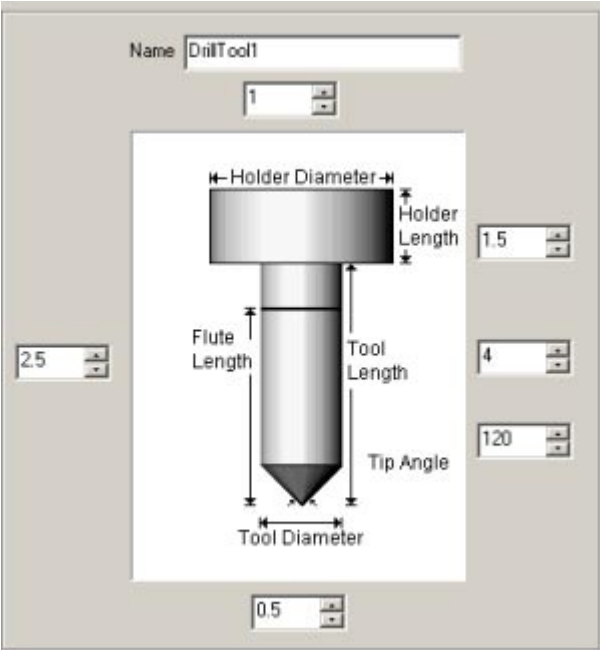
**K Thread Insert****Cut Off Insert (part off)**

Selecting the Drill tool brings up the dialog shown below. Again use the toolbar at the top of the tools dialog to select the desired tool type.

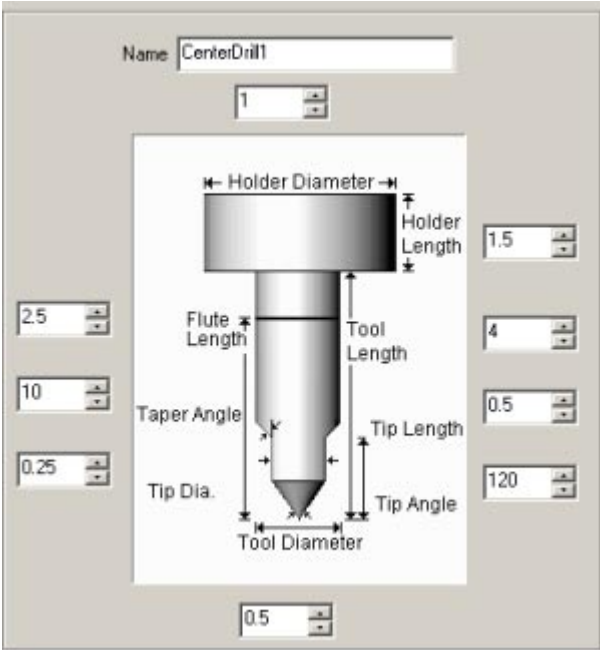


Getting Started with VisualTurn

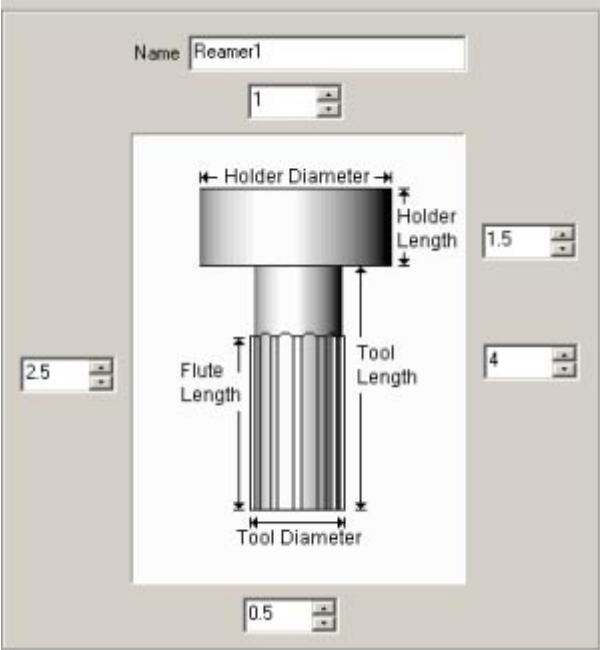
The supported drill tools include:



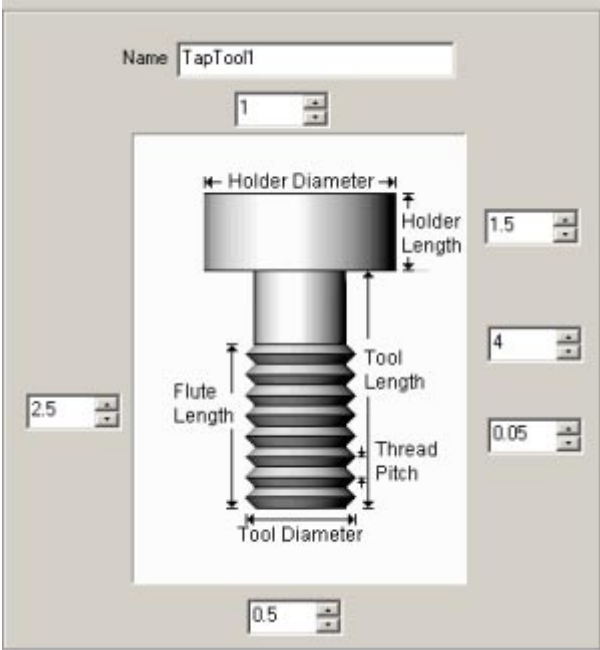
Standard Drill



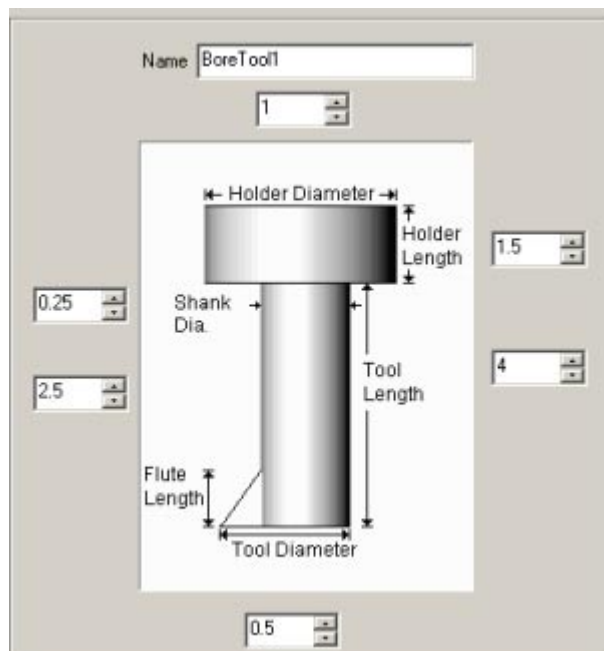
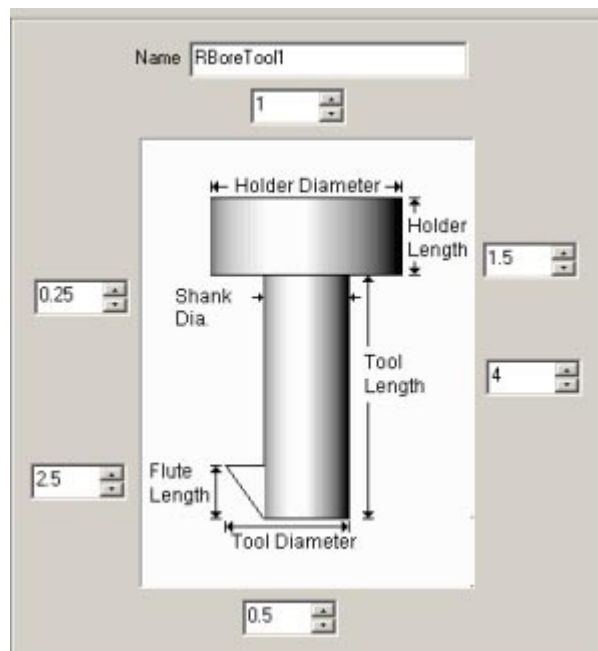
Center Drill Tool



Reamer Tool



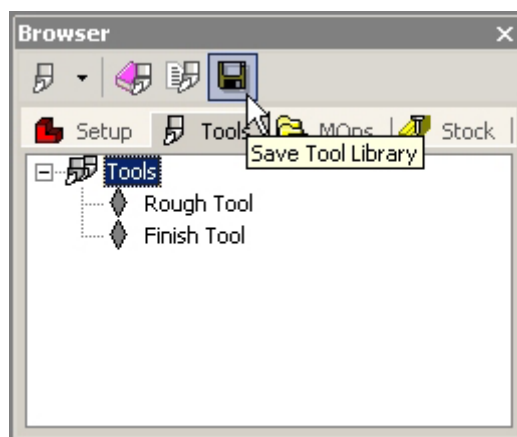
Tap Tool

**Bore Tool****Reverse Bore Tool**

Tool Library

VisualTurn contains two tool library files - **DefaultEnglishTools.vtl** and **DefaultMetricTools.vtl**. These files are located in the Data directory under the VisualTurn installation folder. These files can be used as they are, or you can use them as templates and customize them with your own data. With VisualTurn you can save the tools you create to a library, which can be accessed by future files.

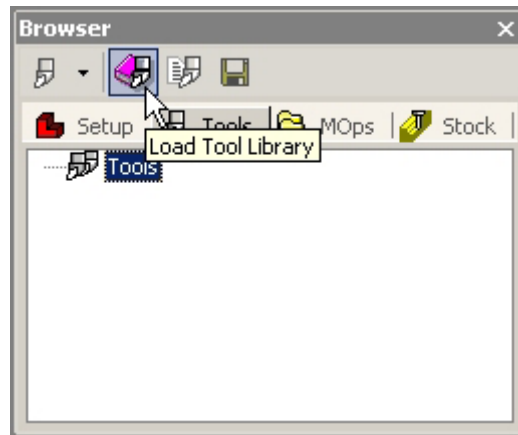
Create/Save Tool library: Once you create a set of tools they can be saved to an external file for future use. Select the Tool / Save Tool Library button in the Tools tab toolbar of the Browser.



Specify a folder location and assign a name. The default extension is *.vtl. Click **Save**.

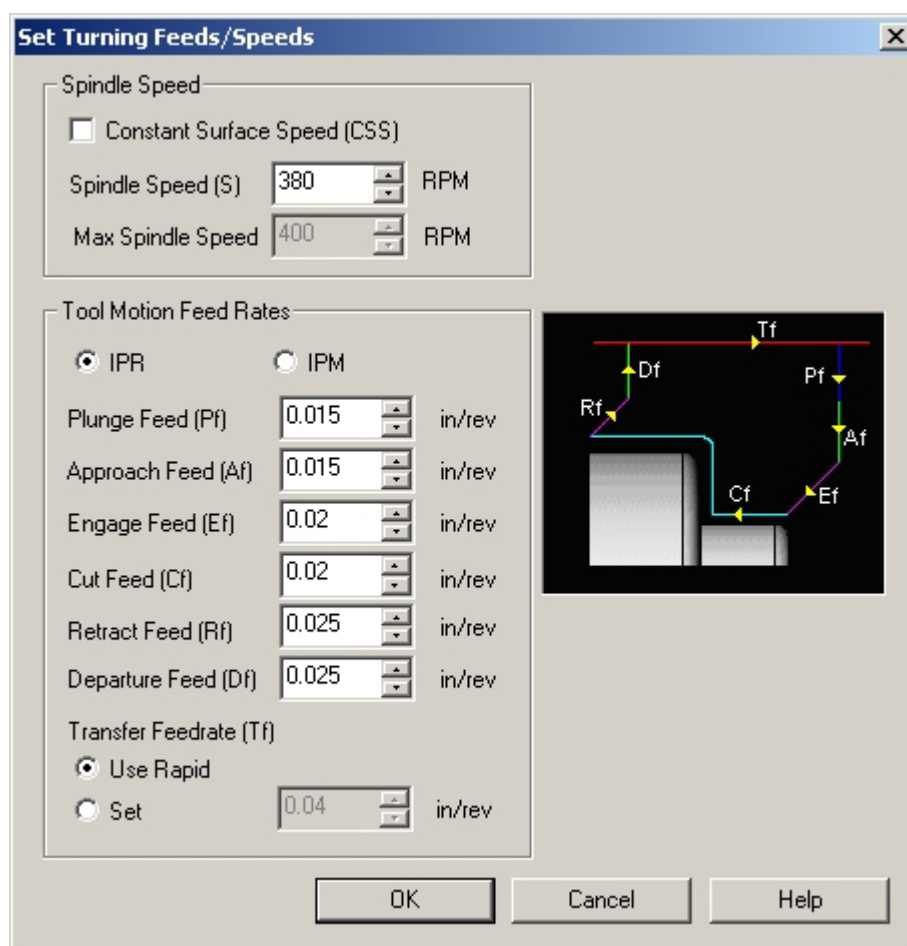
Getting Started with VisualTurn

Load Tool Library: Created tool libraries can be loaded at any time into VisualTurn. To do this select the **Tool / Load Tool Library** button in the **Tools** tab toolbar of the browser. Select the *.vtl file you wish to load.



Feeds and Speeds

You can set feeds and speeds for each operation. You can do this before creating an operation by selecting the Feeds/Speeds dialog and entering in the desired values. Alternatively, once the operation is created you can modify the feeds/speeds associated with the operation. The Feeds/Speeds dialog is shown below.



The various different values that can be set are as follows:

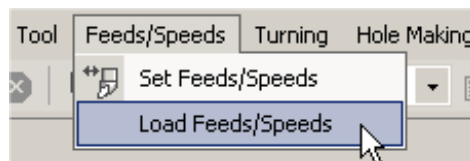
- ❑ **Spindle Speed:** The rotational speed of the spindle, in RPM. If the Constant Surface Speed is turned on, the controller would automatically calculate and adjust the spindle speed based on the current diameter of the work-piece. If this calculated spindle speed is greater than the maximum spindle speed specified, the spindle speed would be reduced to the maximum speed.
- ❑ **Max. Spindle Speed:** The maximum rotational speed of the spindle, in RPM.
- ❑ **Plunge Feed:** The approach feed rate before the tool starts to engage in material.
- ❑ **Approach Feed:** The pre-engage feed rate that prepares the tool just before it starts engaging into material, as it starts cutting. These tool motions are dependent on the machining method.

Getting Started with VisualTurn

- ❑ **Engage Feed:** The feed rate as the tool starts engaging into material. By default this value is 75% of the **Cut Feed**.
- ❑ **Cut Feed:** The feed rate used when the tool is cutting.
- ❑ **Retract Feed:** The feed rate as the tool stops cutting. By default, this is equal to **Engage Feed**.
- ❑ **Departure Feed:** The post-engage feed rate that prepares the tool just as it stops cutting.
- ❑ **Transfer Feedrate:** Specifies the feedrate of transfer motions (air motions). You can either use the **Rapid** setting of the tool, or set a custom feed rate.

Customizing Feeds/Speeds:

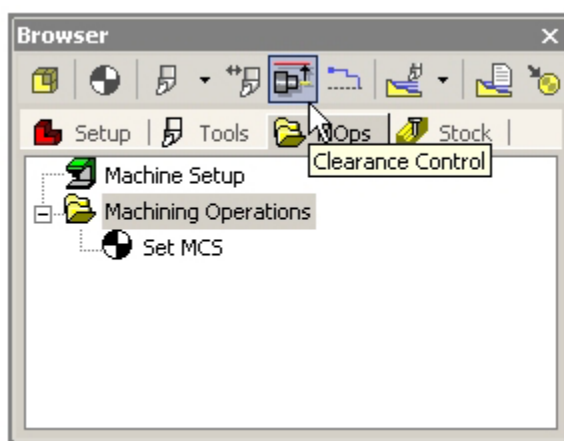
You can also load values from an external table by selecting **Feeds/Speeds / Load Feeds/Speeds** from the dropdown menu at the top.



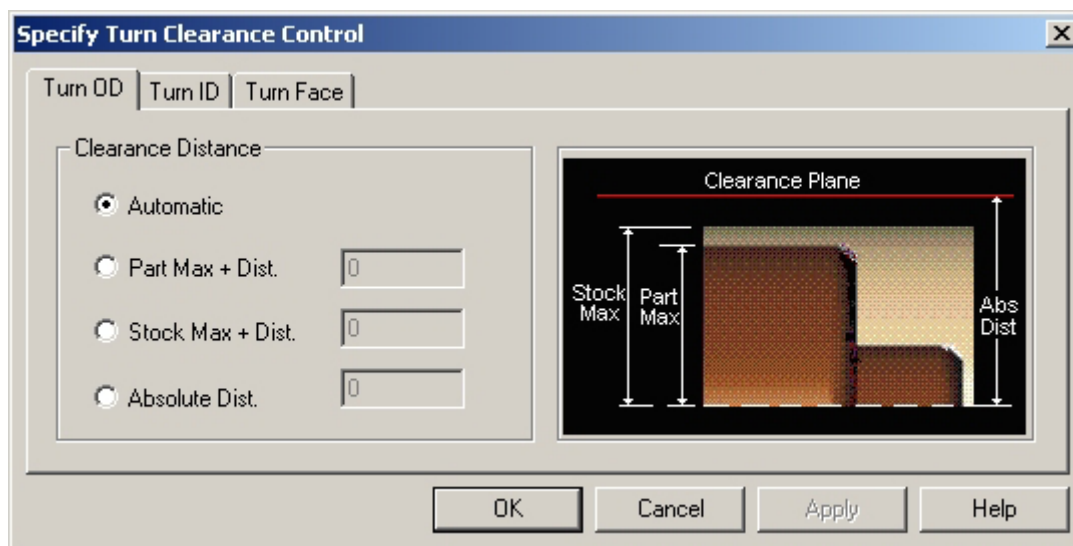
This will load the feeds and speeds from an external text file located in the Data folder under the VisualTurn installation folder. Values for the feeds and speeds can be customized by the user. For more information please refer to the on-line help of the product.

Clearance Plane

The clearance plane is a plane from which the approach motions start and retract motions end. After retracting, the tool moves rapidly along this plane to the position of the next engage. This plane is typically a certain safe distance above the part geometry. The Clearance plane dialog is accessed by clicking the **Clearance Control** button on the **Mops** tab toolbar.



By default (**Automatic** option), the clearance level is calculated by adding a safety distance to the maximum radial point along the approach direction (depending whether Outer Diameter, Inner Diameter or Face is machined) found on both part and stock geometry. This safety distance is set to be the current tool radius. You can set the clearance level to be a specified distance from either the part or stock, or enter the absolute Z level.



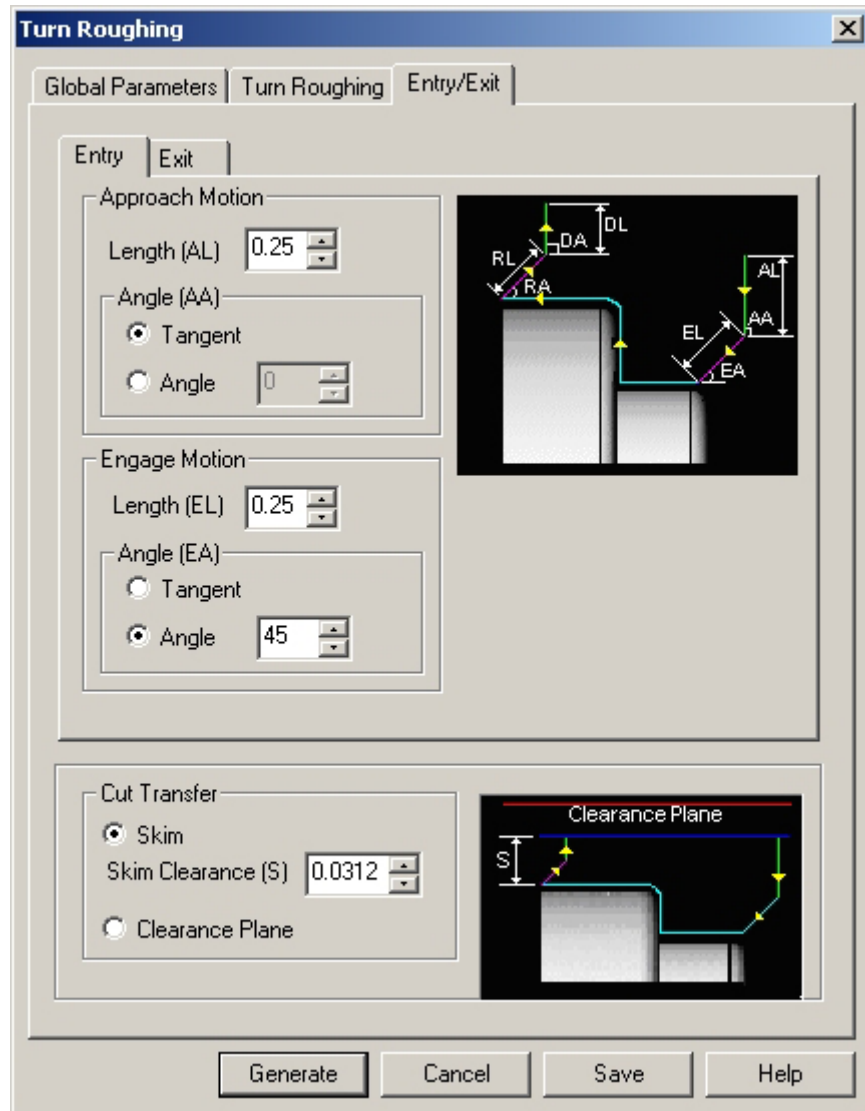
Turn OD Clearance Control

The dialogs for ID and Face approach types are similar. The only difference is that the clearance values are computed along different directions. That is the clearance value will be computed along the negative X axis for ID approach type and along the positive Z axis for Face approach type.

Entry/Exit

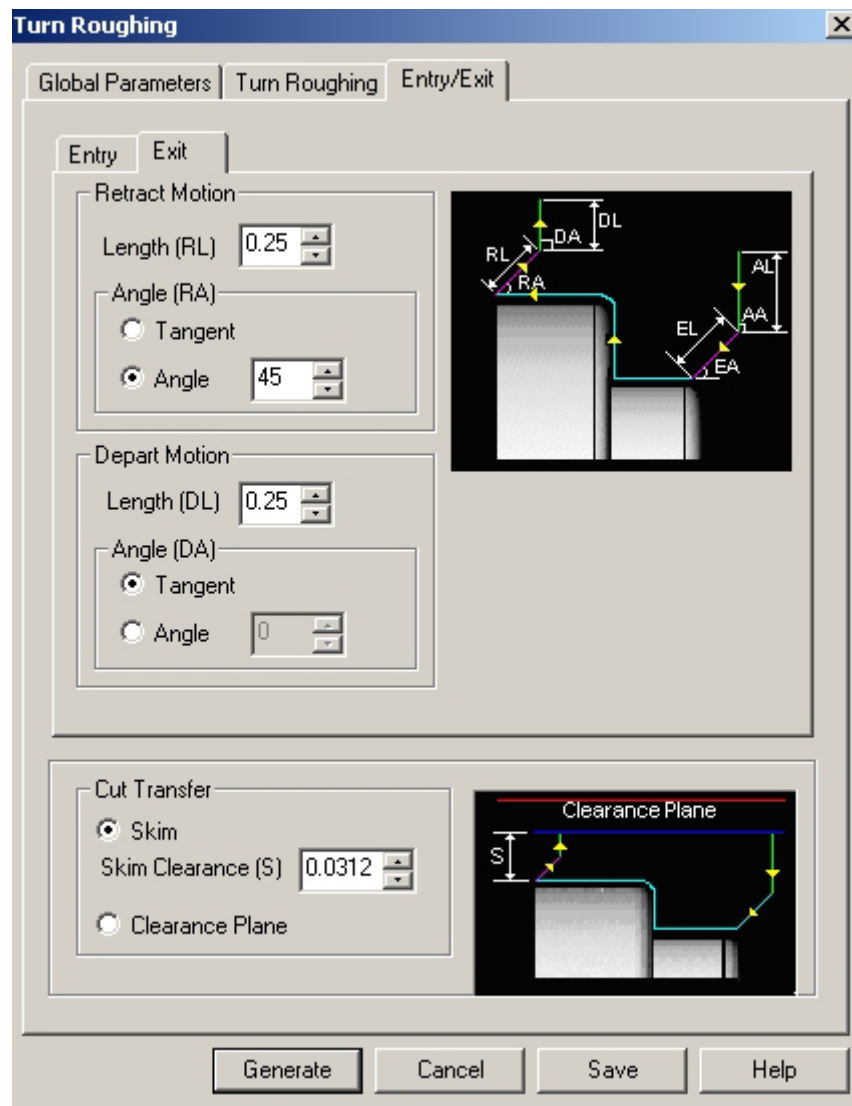
Entry and Exit determines the way in which tool enters and leaves the part geometry. VisualTurn allows the user to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut. The user can also specify the type of transfer motions to perform while cutting.

The Entry motion consists of Approach and Engage. The user can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.



The approach can be either Tangential or at an angle to the Engage motion. This is followed by the engage motion that can be Tangential or at an angle.

Similarly the Exit motion consists of a Retract motion followed by a departure motion. The retract motion can be either Tangential or at an angle. The departure motion can be either Tangential or at an angle to the Retract motion.

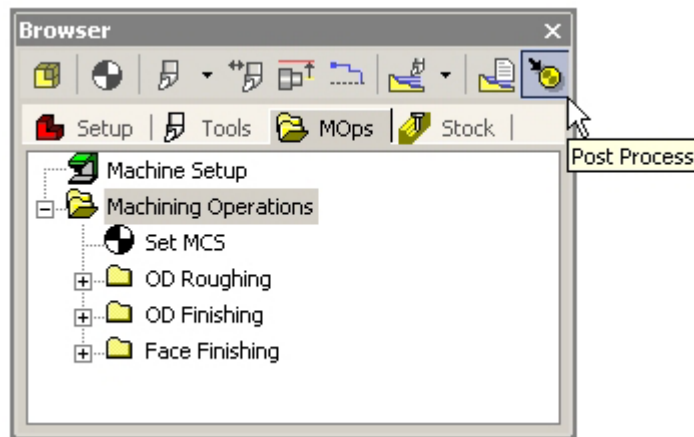


The user can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the regions and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

Post-Processing

Once a machining operation has been generated, it can be post-processed to a specific machine controller. VisualTurn comes with a set of post-processors to choose from. Each post-processor is represented by an *.spm file, all of which are located in the **Posts** folder under the VisualTurn installation folder.

You can post-process an individual toolpath, or all toolpaths at once. For an individual toolpath, right-click on its name in the **Mops** tab of the Browser and select **Post**. You can also click the **Post Process** icon on the **Mops** tab of the Browser.



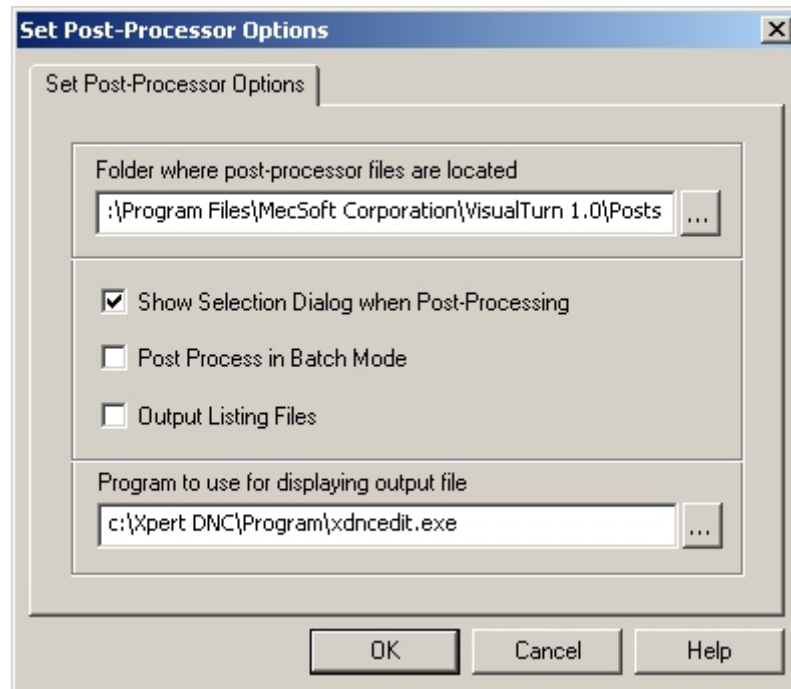
The entire list of toolpaths can be post-processed by right clicking the root folder in the **Mops** tab and selecting **Post All**.

You can also output the toolpath in an APT standard Cutter Location (CL) file. APT is a widely accepted Numerical Control Machine standard. This CL file can then be used to create a machine specific post-processed output through any of the many commercially available APT post-processors.

Post-Processor Problems

If only two built in posts (APT CLS and Roland CAMM GL) are displayed in the selection dialog, then your **Post** folder is not set correctly. Try the following:

1. Select **Post Process / Set Post Options**.
2. Click the **Browse** icon to change the folder where post-processor files are located.
3. Select the **Posts** folder located in the VisualTurn installation folder. (Program Files\MecSoft Corporation\VisualTurn 1.0\Posts)
4. Set the Program to use for displaying output file as notepad or WordPad.



If you are not able post process the toolpath:

5. Under **Post Process / Set Post Options**.

- Make sure **Show Selection dialog when Post Processing** is checked.
- Make sure **Post Process in Batch Mode** is **not** selected.
- Make sure **Output Listing Files** is **not** selected.

Post the machining operations, making sure you are browsing to the **Post** folder in the VisualTurn installation folder. For the output file at the bottom, make sure there is a valid file name (valid path).

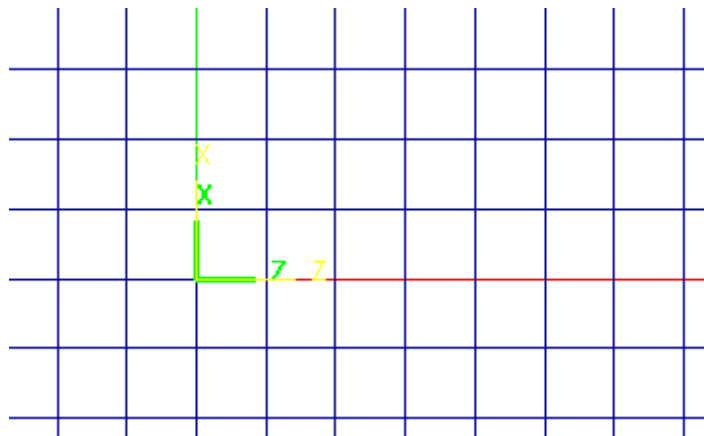
CAD Tutorial for creating a 2D profile for Turning

As you have seen, you can import a ready-made part into VisualTurn. If you want to create your own part from scratch from within VisualTurn, the **Geometry Bar** contains all the CAD tools you need.

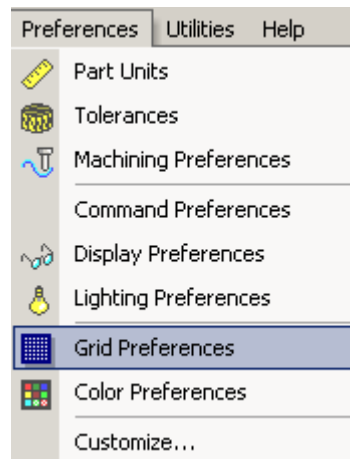


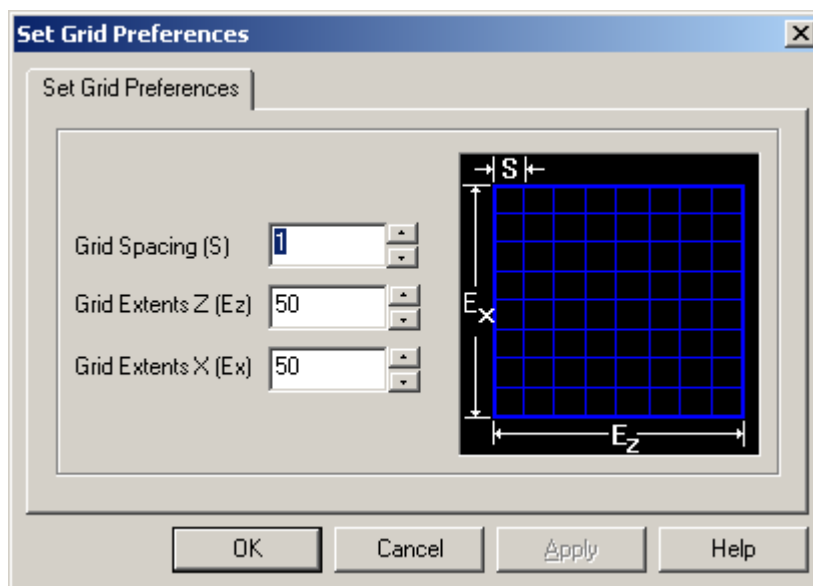
To set up the grid:

1. Start a new file, switch to Top view, and display the grid. The default grid spacing, assuming you are working in inches, is 1”.



2. If you wish to change the grid settings, select **Preferences / Grid Preferences** to edit the grid spacing and grid extents. Choose grid spacing to 1.0 for this tutorial.



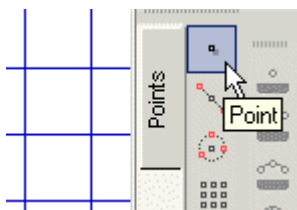


3. To create geometry with respect to the grid, you must be able to snap to grid points. In the lower right part of the screen, make sure **Grid Snap** is activated.



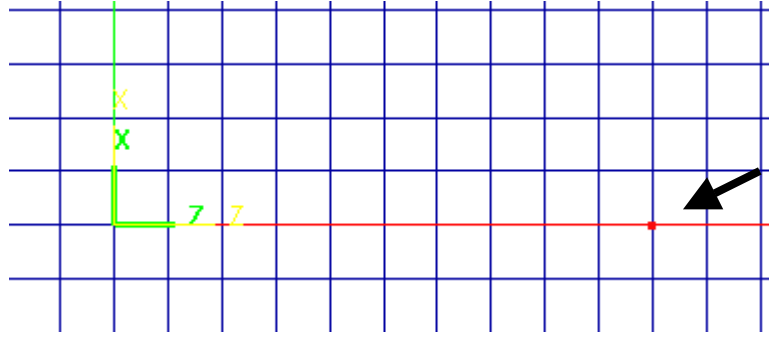
To create point regions:

1. As you've already seen in previous exercises, all geometry tools are in the **Geometry Bar**, located by default on the right side of the screen. In the **Points** category, click **Point**.



2. Place the first point at 10" to the right of origin. This is ten grid lines away, or you can look at the cursor location indicator at the lower right corner.

Getting Started with VisualTurn

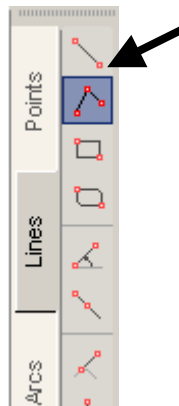


3. Hide the grid and all coordinate systems, and you should be able to see the point clearly.



To create a part profile (Region):

1. Set to the Top View
2. Draw a 2D profile of the part using the CAD tools available from the geometry bar to the right of your screen.
 - a. For Example: Switch to the **Lines** category and select **Polyline**.

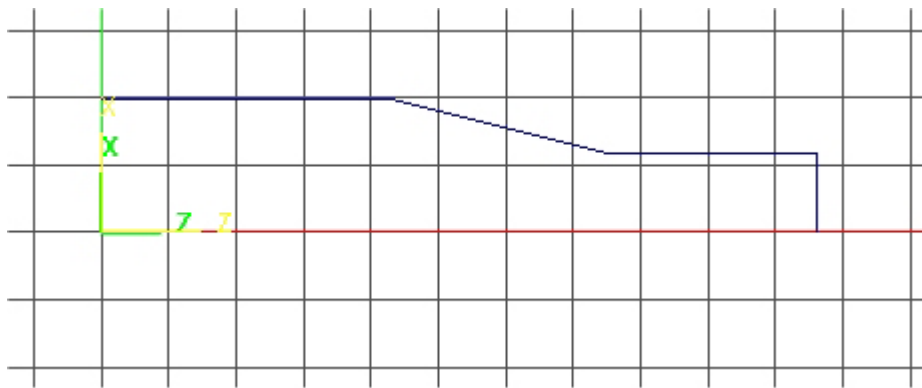


- b. While the Polyline mode is active, start with the origin and choose the points of the part profile in succession. This would start building the part profile. Right-Click to indicate the end of the polyline.

- c. You may use a combination of lines, polylines, and arcs to create geometry.
3. Make sure the 2D profile be closed (touches X and Z axes) in the First Quadrant of the lathe coordinate system.
4. Use the chain/join tool from the edit curves tab on the Geometry toolbar to join 2 or more lines/curves.



5. The part is now ready for programming



Example of a 2D profile created in VisualTurn

6. The above **2D profile** can be selected as a **region** for creating turning operations.
7. Region for Hole Machining Operations
 - a. A point region to indicate the starting location of the drilled hole or
 - b. The above 2D profile can be select to create a drilled hole. (*Visual Turn analyzes selected the 2D curve and determines the drill point where the 2D curve intersects the Z axis at $X=0$*)

Creating Surfaces

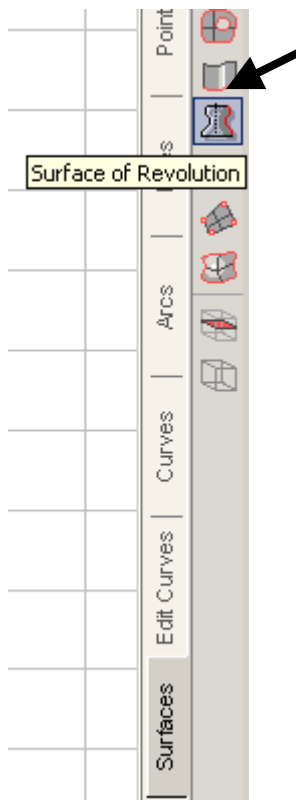
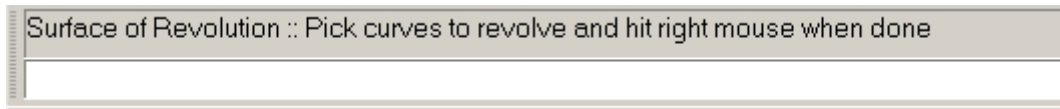
In this final section, the curves will be used to create the surfaces of the part.



To create the revolved surface:

1. Select the 2D curve created from the above example.
2. Click on **Surface of Revolution** surfaces and you would be prompted to enter/select the start point of the axis of revolution.

*Note: If no curves are selected and you pick **Surface of Revolution**, VisualTurn prompts you to select a 2D curve and right click the mouse button when the selection is complete.*

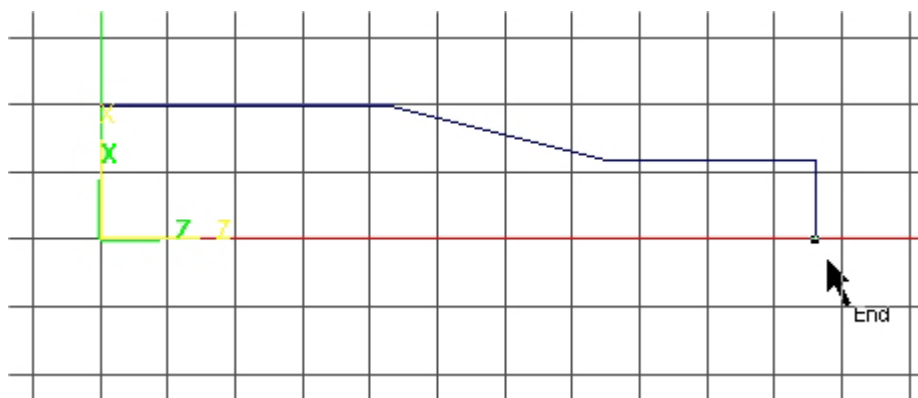


Surface of Revolution :: Pick start point of axis of revolution or enter coordinates x,y and z

- Pick the origin point as the start point and you would be prompted to enter/select the end point of axis of revolution.

Surface of Revolution :: Pick end point of axis of revolution or enter coordinates x,y and z

- Select the end point (see below)



- Enter the start angle as 0.0 and end angle as 360.

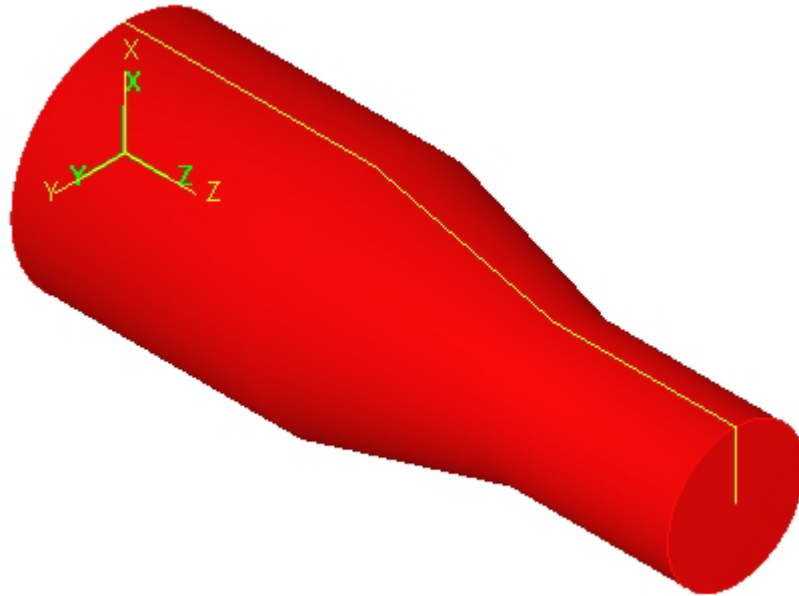
Surface of Revolution :: Enter start angle of revolution::[sA=0.000000]

0

Surface of Revolution :: Enter end angle of revolution::[eA=45.000000]

360

- Hit **Enter** and a surface of revolution is created.

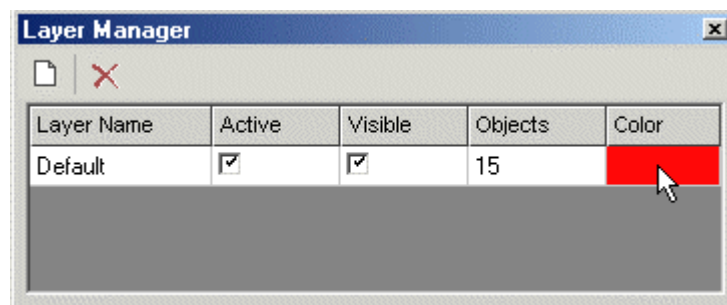


To change display settings:

7. The part is located on the Default layer. If you want to change the color of the part, you must change the color of this layer. Click the **Layers** icon.



4. In the Layer Manager, click the **Color** box to select a new color for the part.



Tutorial 1: Roughing & Finishing

In this tutorial, you learn to create roughing and finishing toolpaths to program a designed part.

The stepped instructions are accompanied by explanatory and introductory text. Reading this text will help you understand the tutorial methodology and provide information about additional options available. However, if you prefer to work straight through the steps without any additional reading, look for the following symbol:



Don't forget to save your work periodically! You may want to save the file under a different name so that the original file will be preserved.

Loading a Part Model

“Part” refers to the geometry that represents the final manufactured product. You can create parts within VisualTurn, but it is more typical to import geometry created in another CAD system.

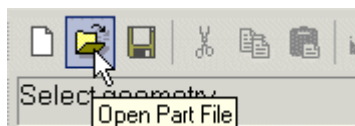
You can import solid models of Stereo-Lithography (both ASCII and binary) format files. Surfaces can be imported from IGES or Rhino 3DM. Faceted (triangulated) models can be imported from VRML, Raw Triangle, DXF / DWG facet data, or Rhino Mesh. Non-faceted geometry, once imported, is immediately converted and stored as triangulated data.

Imported geometry is stored internally as a VisualTurn part file. This allows for much faster part loading time.



To load a part:

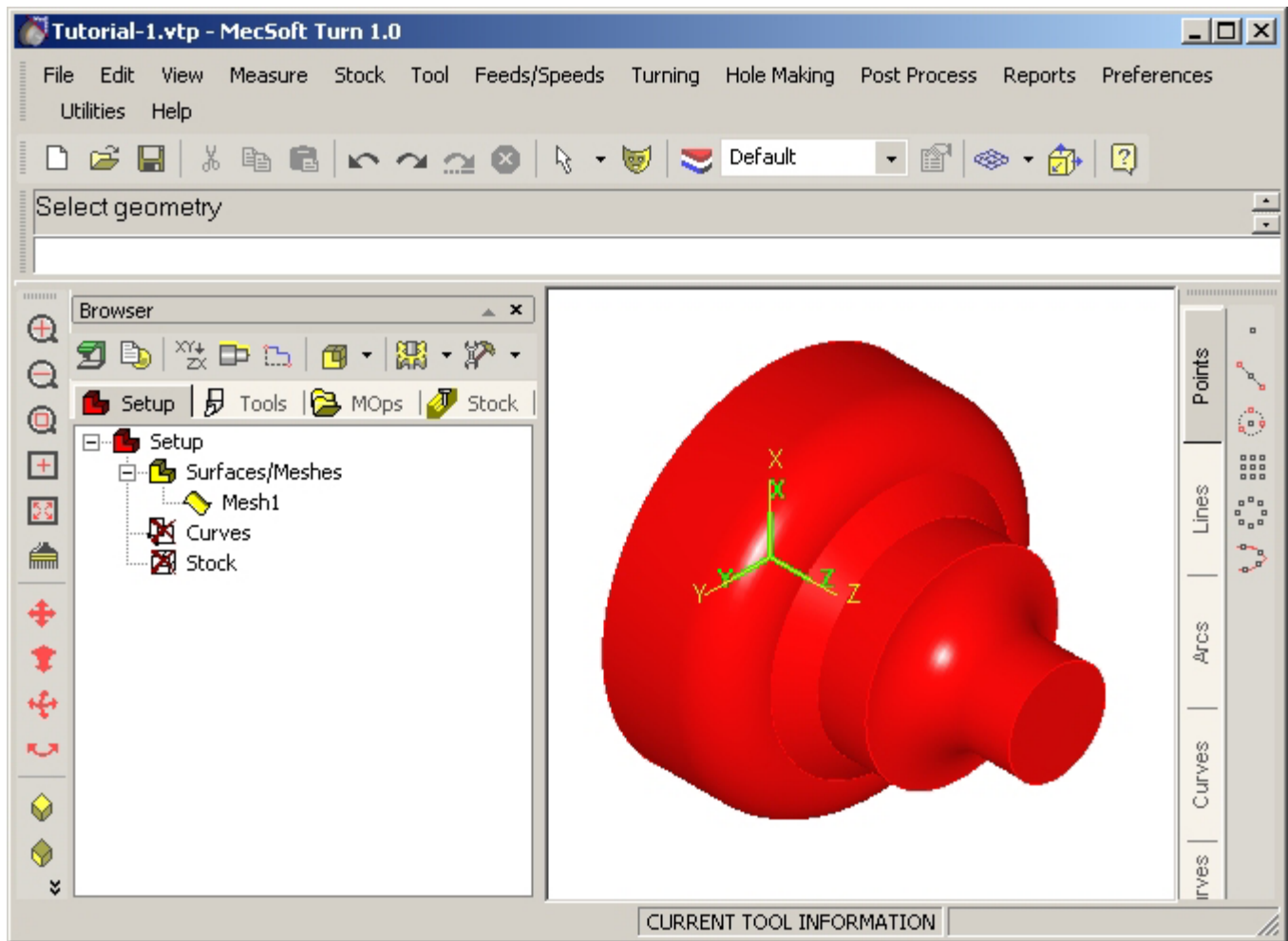
1. Select **File / Open**, or click the **Open Part File** icon from the **Standard** bar.



2. From the **Open** dialog box, select the **Tutorial-1.vtp** file from the **Tutorials** folder in the VisualTurn installation folder

The imported part appears as shown below.

Getting Started with VisualTurn



VisualTurn also allows you

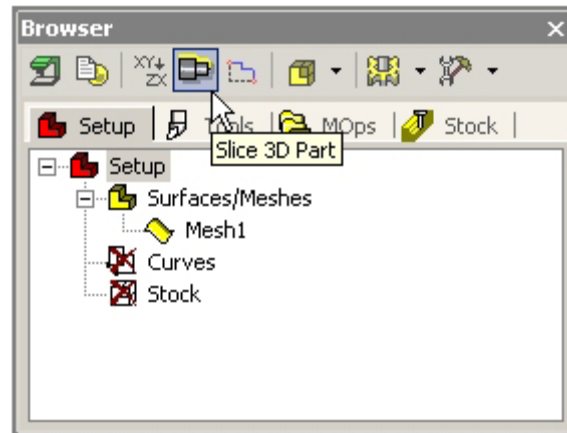
- Create 2D profiles using the CAD features
- Import 3D CAD files in standard format. However, these 3D files have to be sliced to reduce them to 2D profiles.

***Note:** Refer to our CAD section for help on creating 2D profiles and other CAD tools*

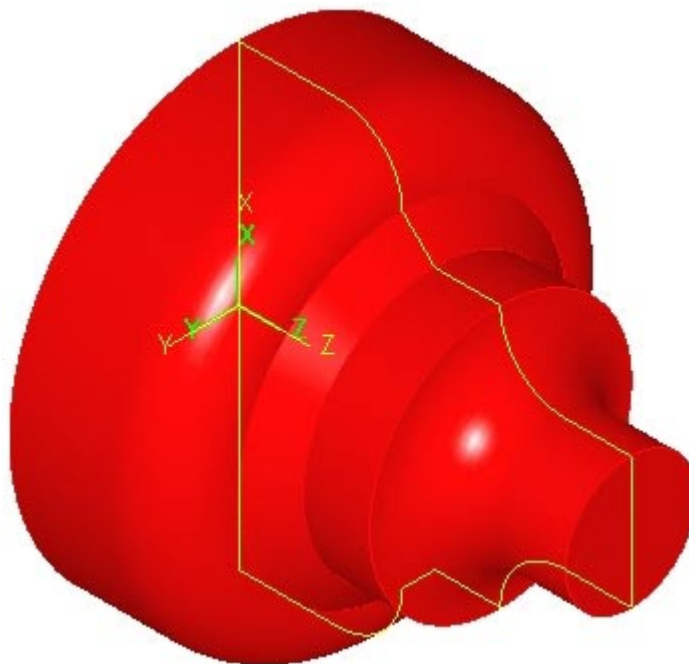


To slice a part and resolve part region

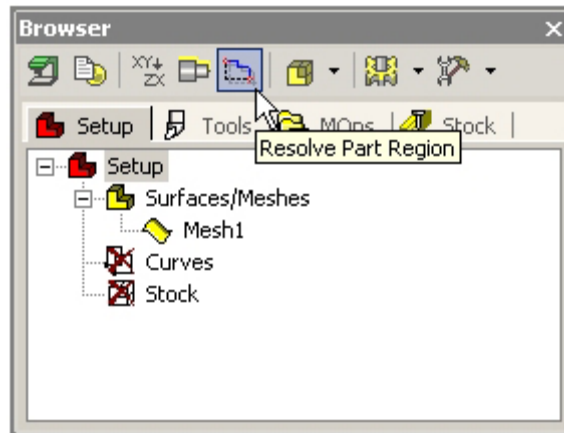
3. Once the part is loaded click on the **Slice 3D Part** on the Setup bar of the Browser.



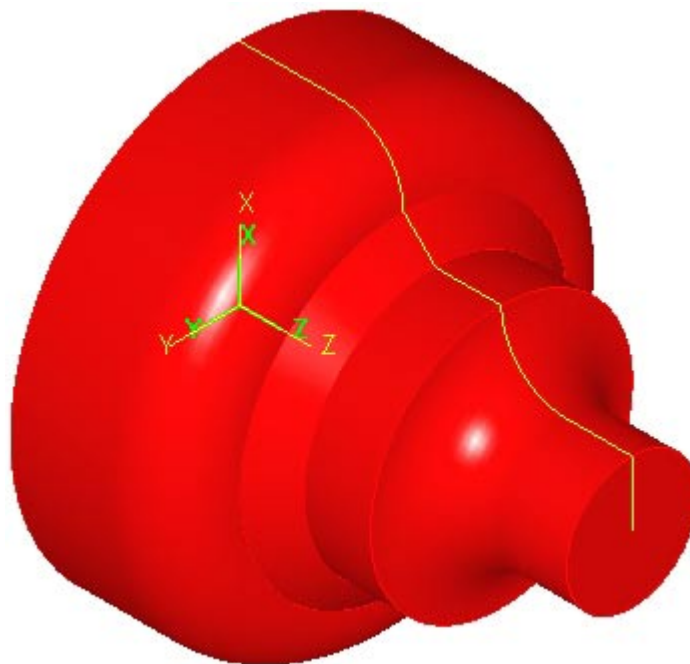
4. This slices the 3D file into a 2D profile.



3. Select the 2D profile and click Resolve part region to extract the curves to First quadrant of the lathe coordinate system (X and Z axes)



4. The resolved 2D curve appears as shown below

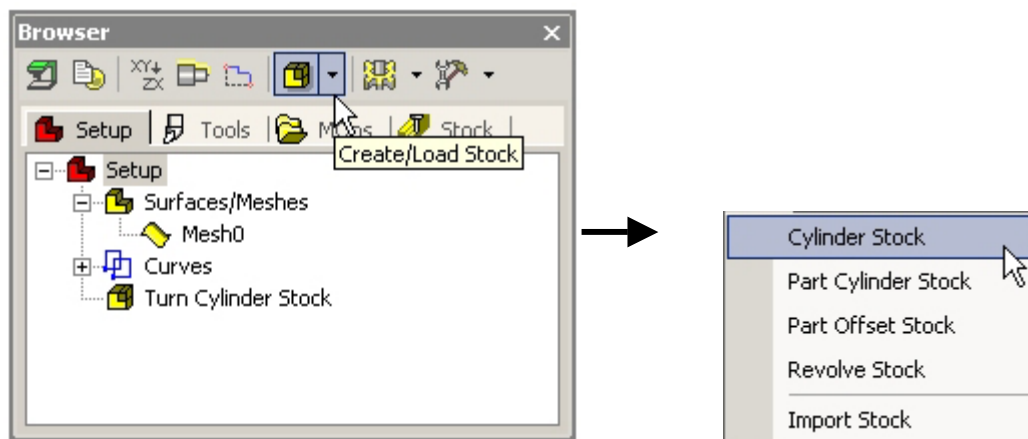


5. You may now save the file and start creating VisualTurn machining operations.

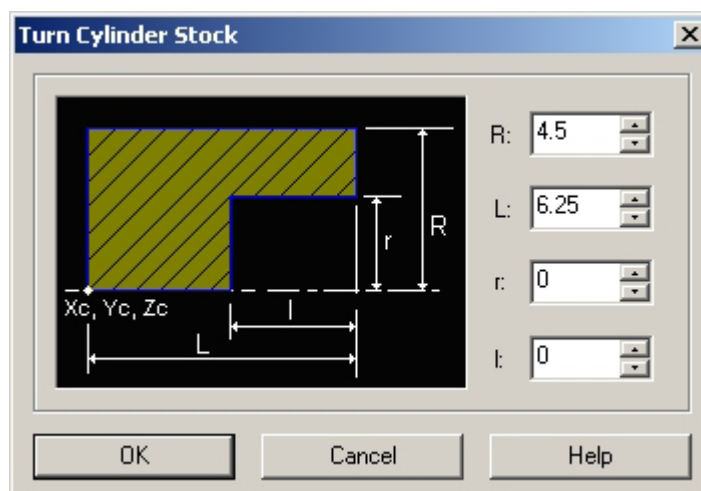


To create the stock:

1. Select **Create/Load Stock** from the **Setup** tab of the Browser and select **Cylinder Stock**. (This tool is also available on the **Stock** tab of the Browser.)

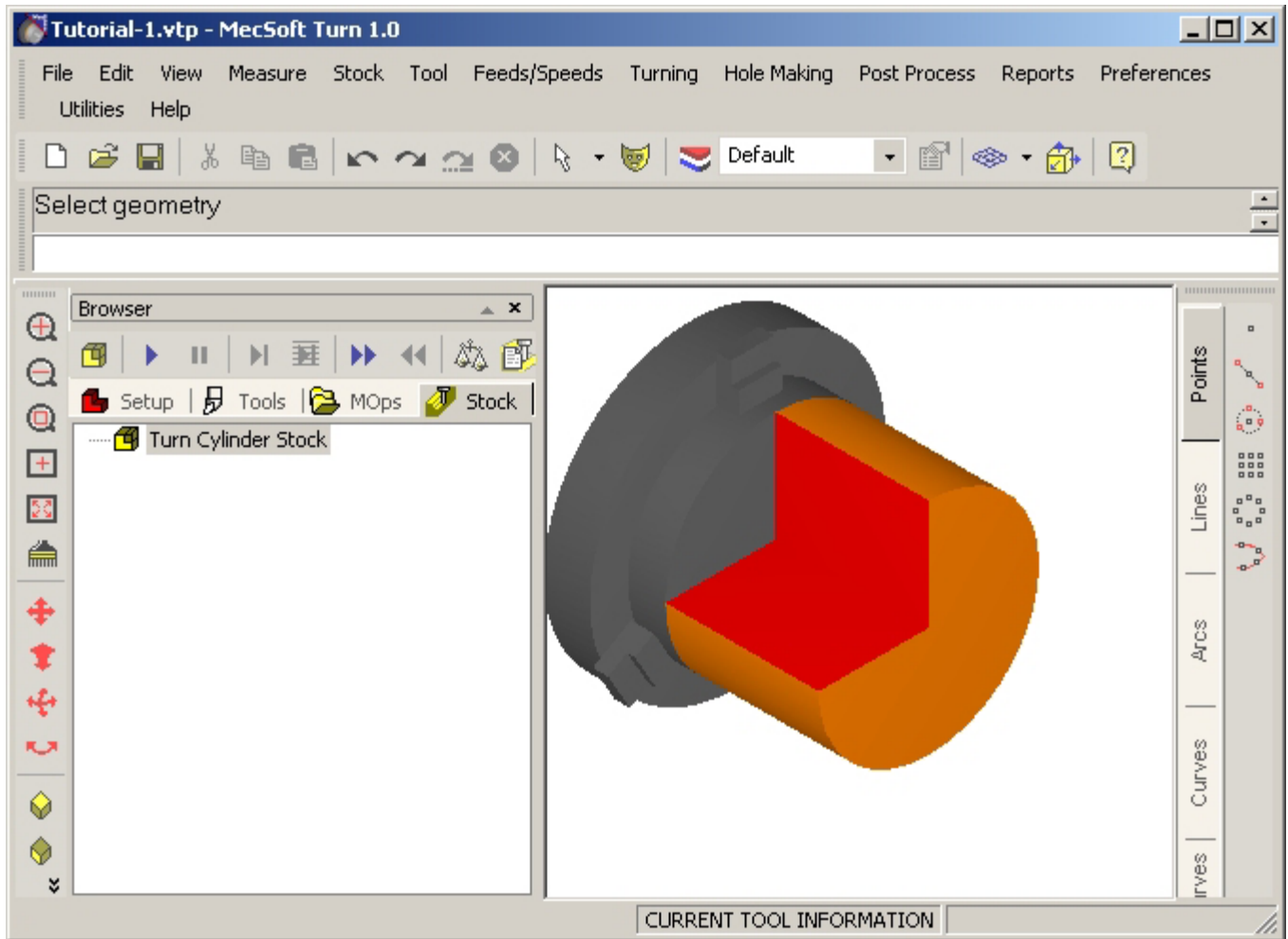


2. In the **Cylinder Stock** window, you can enter the radius and length of the stock. Enter the values as shown in the illustration below. Click **OK**.



3. The stock model is created. To display the stock, click the **Stock** tab of the Browser. The stock is displayed as a cylinder positioned at the reference point of the lathe machine. Its color can be set in the **Color Preferences**.

Getting Started with VisualTurn



(The simulation settings are set to 3-quarter view)

4. If you don't see the stock, make sure the **Hide Stock** toggle icon in the **View** bar is not pressed.



5. To change simulation setting click on the Simulation Settings on the Stock tab. For more help look under simulations settings in the user manual.

Tip: Stock is used for simulation, and its display involves data-intensive rendering. This can slow down VisualTurn's performance. Therefore, we recommend turning off the stock display when not needed.

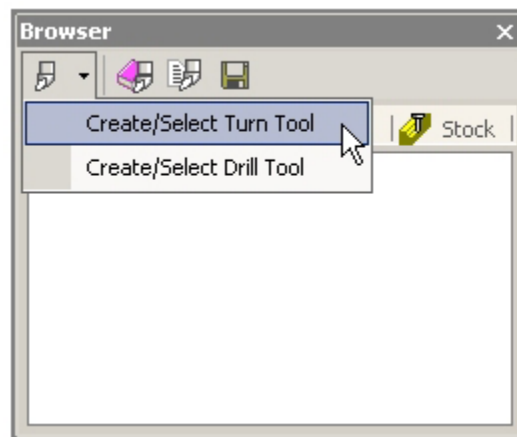
Note: You must define a stock model before creating Roughing and Groove Roughing operations. All other operations can be created without first creating a stock model.

Creating Tools

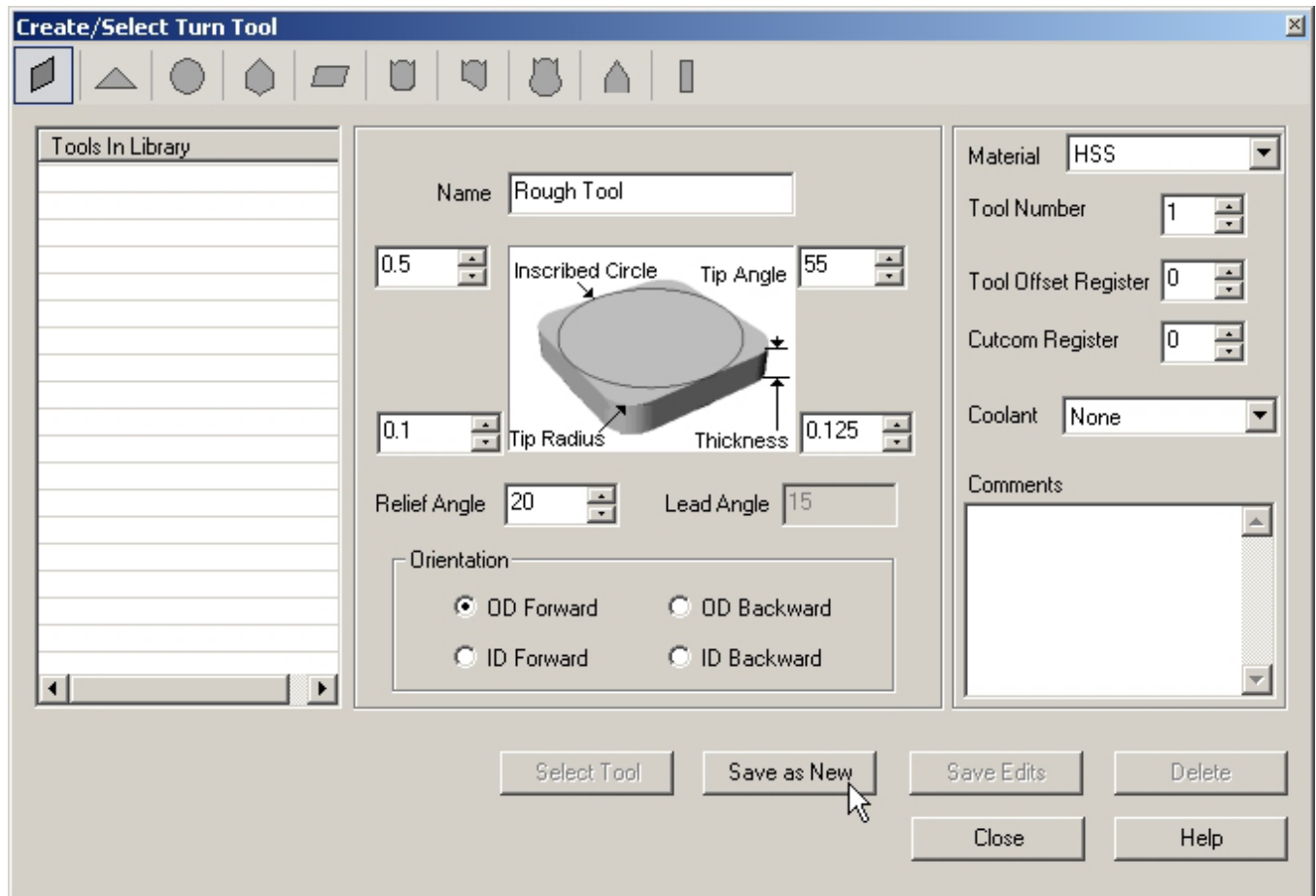


To create the roughing tool:

1. Select **Tool / Create/Select Turn Tool**, or click the **Create/Select Turn Tool** icon on the **Tools** tab of the Browser.



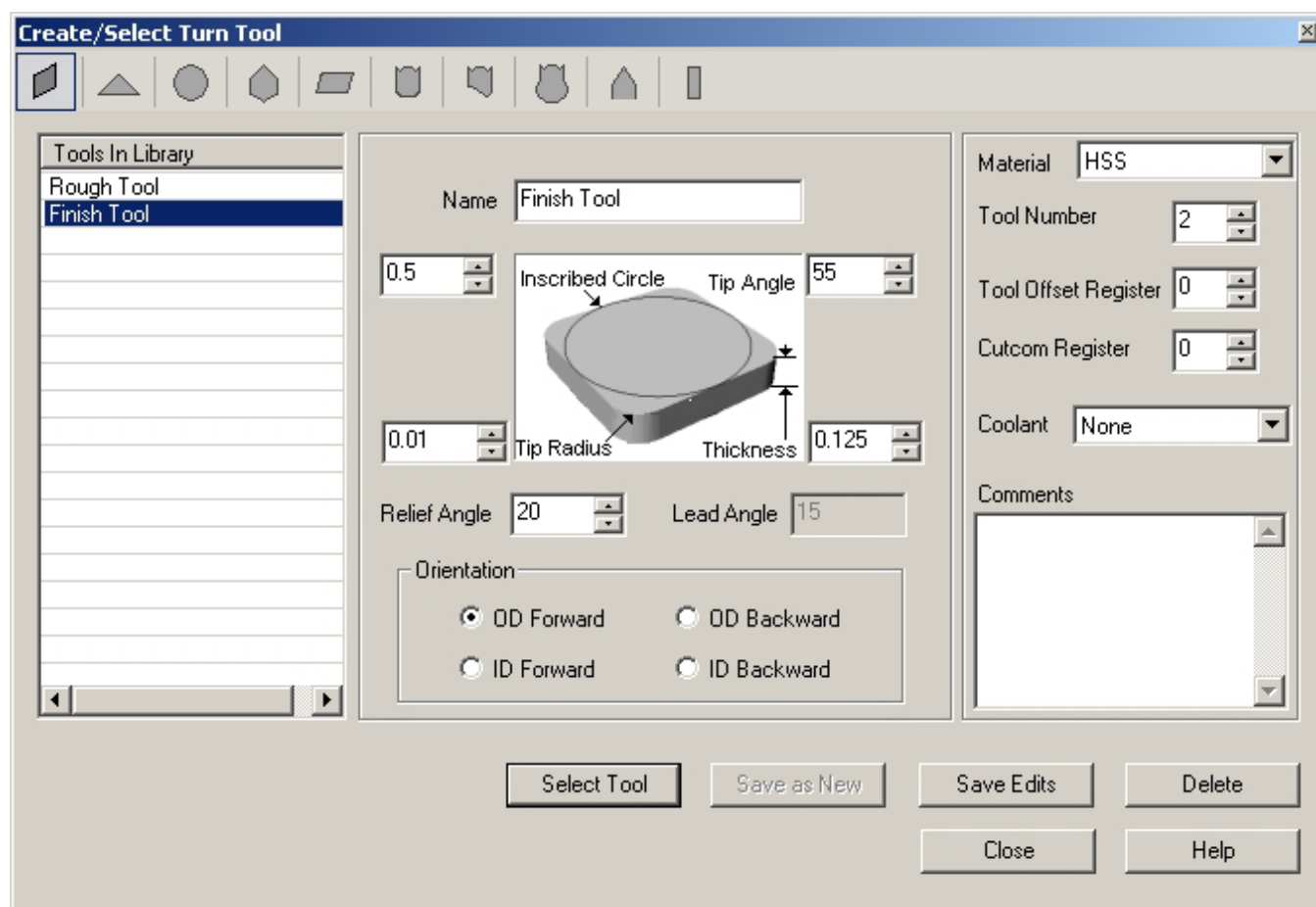
2. In the **Select/Create Turn Tool** window, click the **Diamond** tab. Change the name to **Rough Tool** and Tip Radius to 0.1". Choose the default values for other parameters and click **Save as New**.



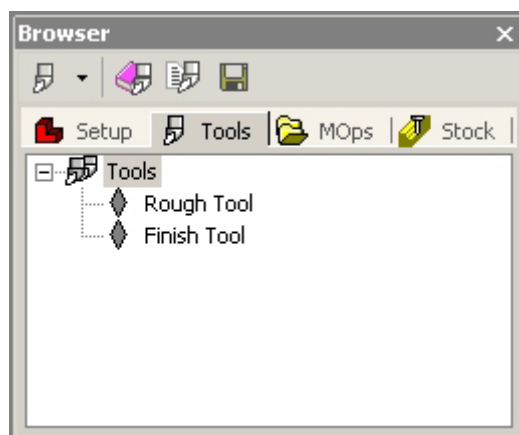
To create the finishing tool:

Finishing is typically performed with a smaller radius tool.

1. While still in the **Diamond** tab, change the tool diameter to 0.01 inches.
2. Change the tool name to **Finish Tool**.
3. Click **Save as New**.



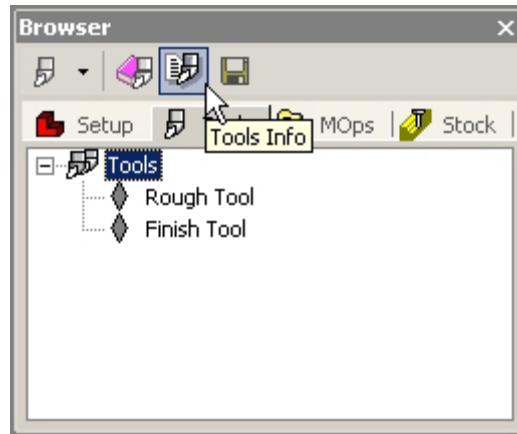
4. Click **Close** to close the window.
5. Now that all tools have been created, click the **Tools** tab in the Browser. All the tools are listed.



Note: You can double-click on any tool to open its definition window. This is an easy way to make changes, if needed.

To see the information of all tools, click on the Tools Info icon or right-click on the **Tools** header and select **Information** or click on the Tools Info from the tools tab.

Getting Started with VisualTurn



This displays a table listing the properties of all the tools you've defined.

The screenshot shows a 'Turn Tool Information' dialog box. It contains a table with 9 columns: Name, Tool Type, Orientation, Radius, Nose Angle, Relief Angle, Lead Angle, Tool #, and Material. The table has two rows of data: 'Rough Tool' and 'Finish Tool'. Below the table is a 'Print' button, and at the bottom are 'OK', 'Cancel', and 'Help' buttons.

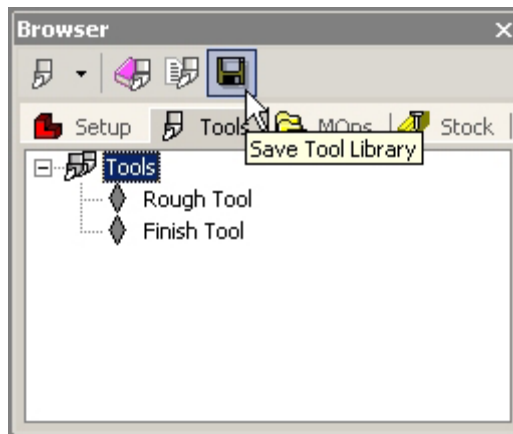
Name	Tool Type	Orientation	Radius	Nose Angle	Relief Angle	Lead Angle	Tool #	Material
Rough Tool	TurnTool	0 deg	0.1000 in	55.0000 deg	20.0000 deg	15.0000 deg	1	HSS
Finish Tool	TurnTool	0 deg	0.0100 in	55.0000 deg	20.0000 deg	15.0000 deg	2	HSS



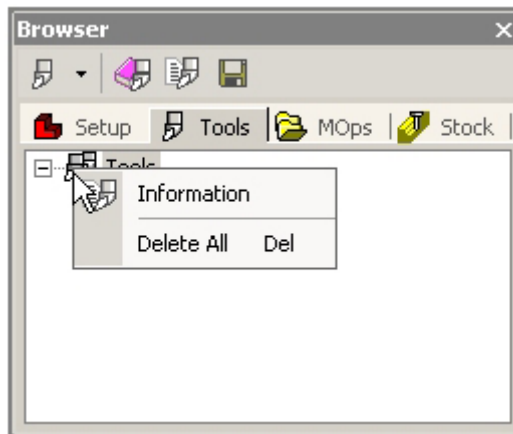
To create the tool library:

You can save the tools in the list to a library, which can be accessed by future files.

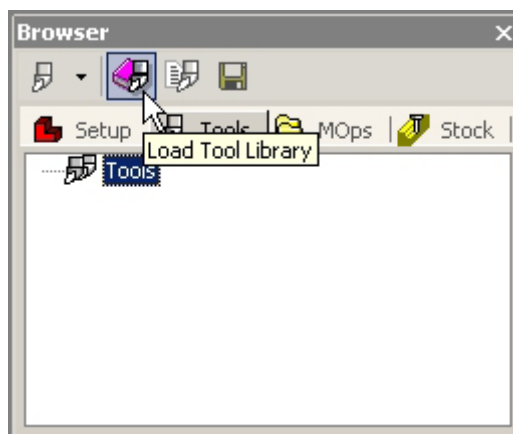
1. A group of tools can be saved to a library file for future use. Select **Tool / Save Tool Library** or click the icon in the **Tools** tab of the Browser.



2. In the default folder (should be **Tutorials**, which contains the part file), assign the name **OD_Turn Tools**. The default extension is *.vtl. Click **Save**.
3. Right-click on the **Tools** header in the Browser and select **Delete All**.



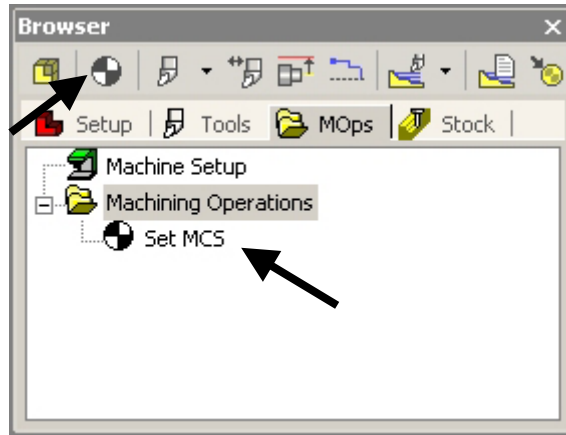
4. To replace the tools, select **Tool / Load Tool Library**. Select the *.vtl file you just saved, and the tools reappear in the file.



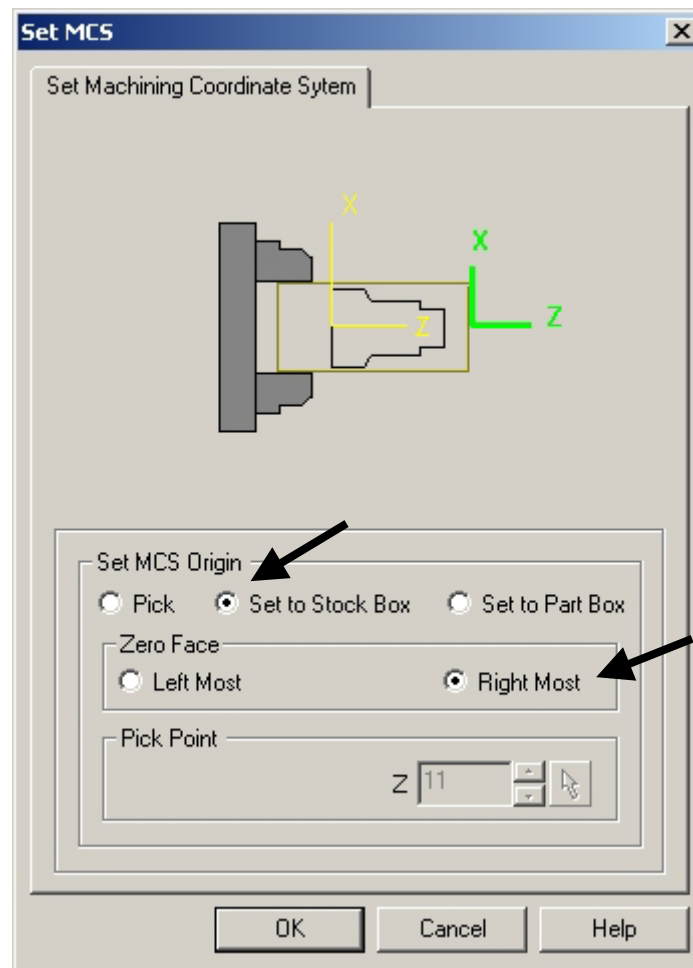


To set the Machine Co-ordinate System:

5. Click on the **Set MCS** icon in the **Mops** tab of the Browser Bar or Double-Click on the Set MCS under Machining Operations

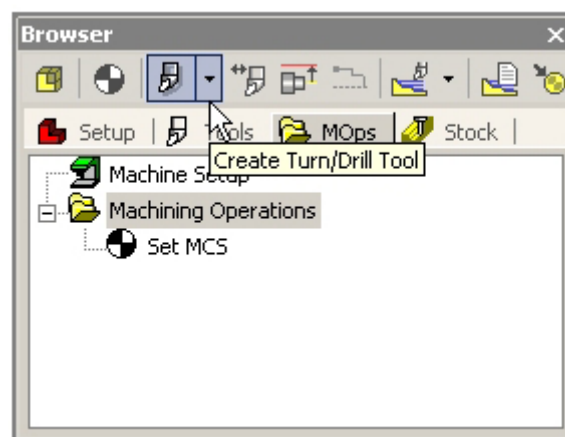


6. This gives the user different options to set the machine zero. The user can set the zero to the left/right face of the part/stock box or pick the point directly.
7. For this example, set the **MCS** origin to the **Stock Box** and Zero Face to **Right Most** face of the stock. Click **OK**.



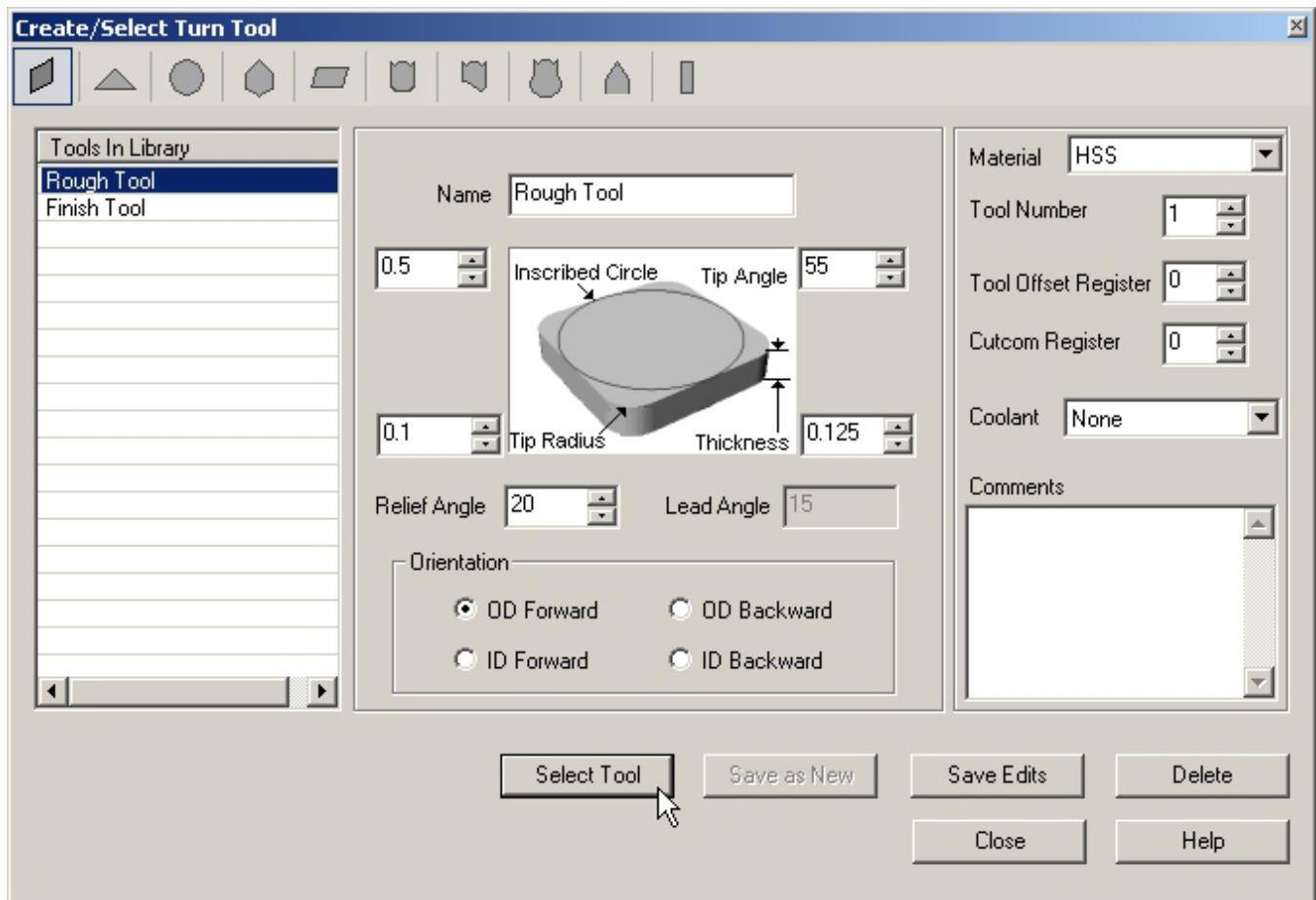
Selecting the Tool for Roughing Operation

1. Under the Mops tab click on Create/Select Turn tool



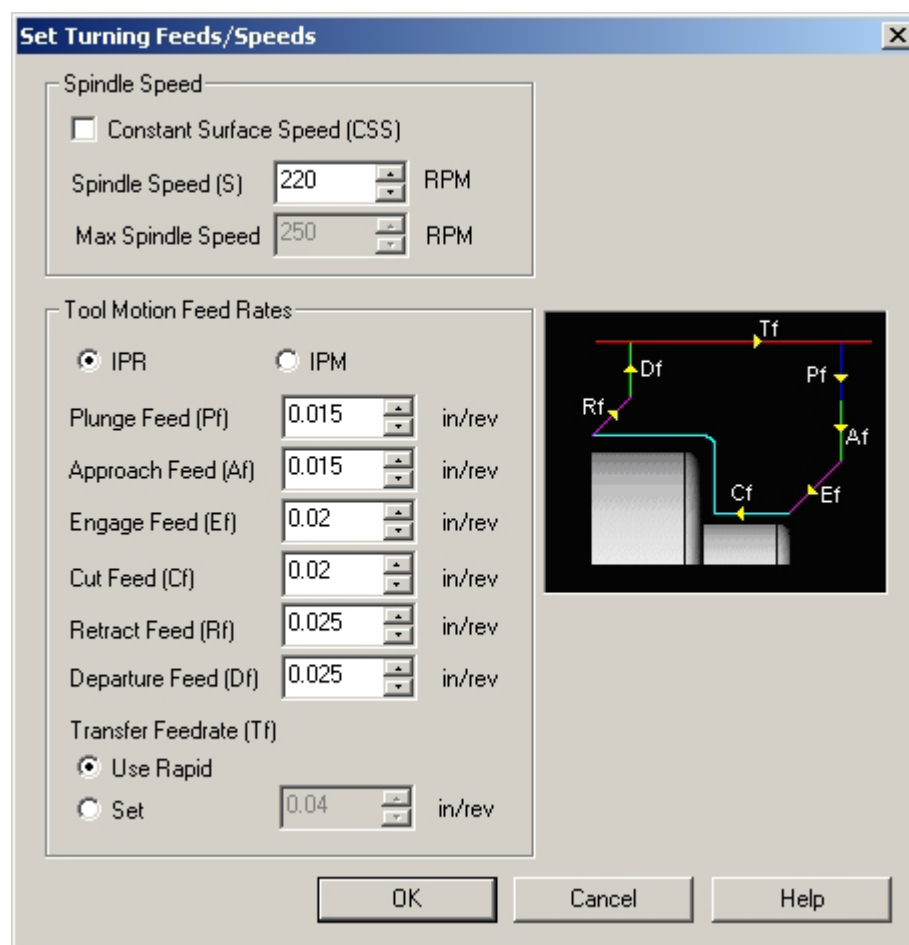
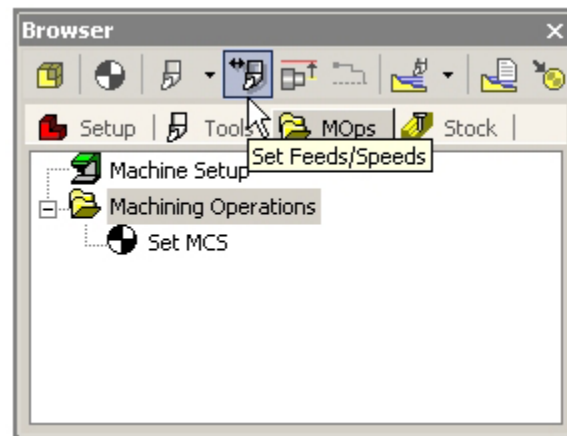
Getting Started with VisualTurn

2. From the tool select dialog pick the Rough tool and click select tool. This will make the Rough tool as the active tool and shows up in the status bar at the bottom of the screen



Setting Feeds and Speeds

You can set toolpath feeds and speeds and customize these settings for later use. To set the feeds and speeds click on Set Feeds/Speeds on the Mops tab. This launches the Feeds/Speeds dialog. Considering the Stock material as Aluminum and the Tool material to be HSS for the above example.



Once you have set the Speeds and Feeds click OK to continue.

These feeds and speeds will be used during the post-processing of the toolpath.

Creating the Outer Diameter Roughing Toolpath

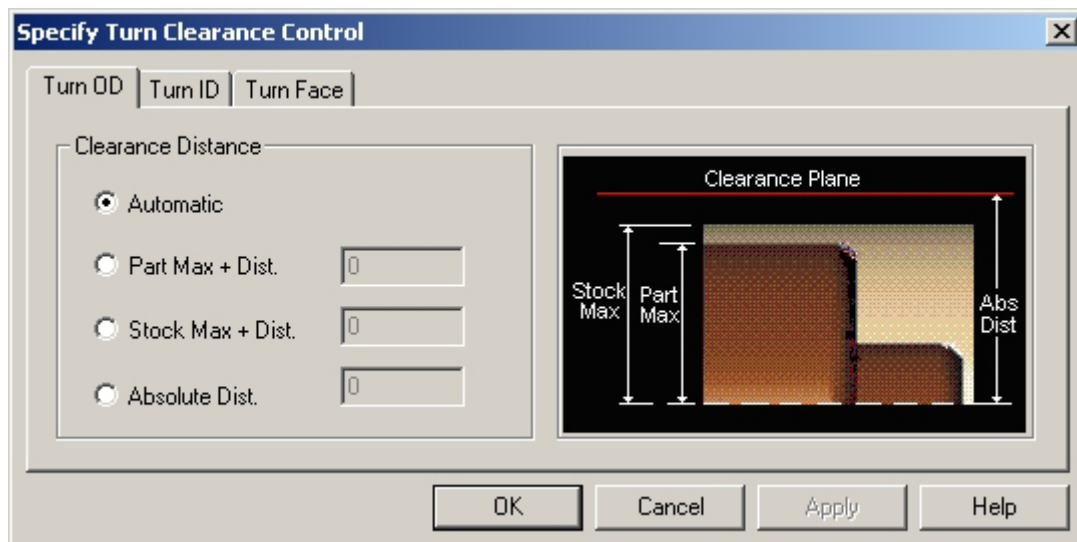
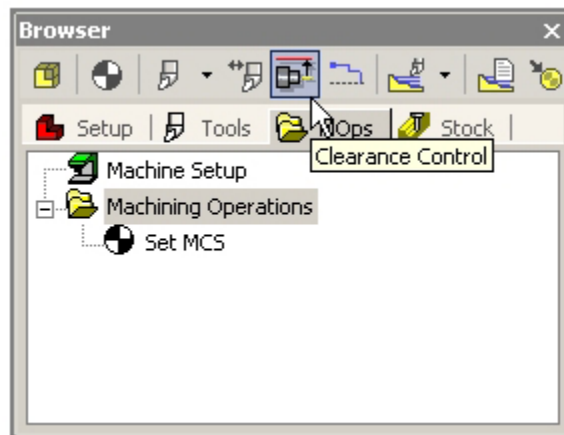
In this type of toolpath, VisualTurn uses stock geometry and part geometry to determine the machining region. The safe machining region is the region in which the tool can safely traverse removing stock. Once this machining region is determined, a tool traversal pattern such as a zigzag or offset machining cut pattern can then be applied to remove stock.

Regions are curves that already exist in the model, or curves you create within VisualTurn. In the **Setup** tab of the Browser, you will see that one region already exists in this file.



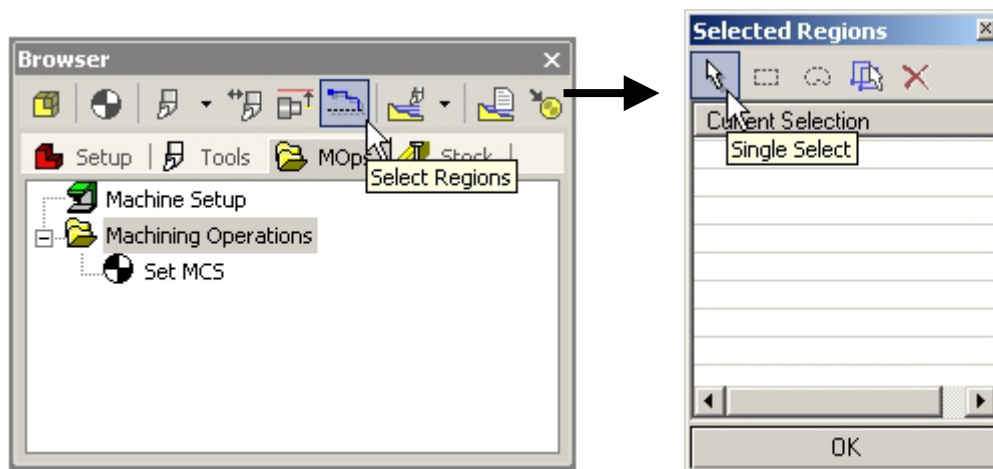
Setting Clearance Plane

The clearance plane is a plane from which the approach motions start and retract motions end. After retracting, the tool moves rapidly along this plane to the position of the next engage. This plane is typically a certain safe distance above the part geometry. Clicking the **Clearance Control** button on the **Mops** tab sets clearance levels. By default the Clearance Distance is set to automatic.

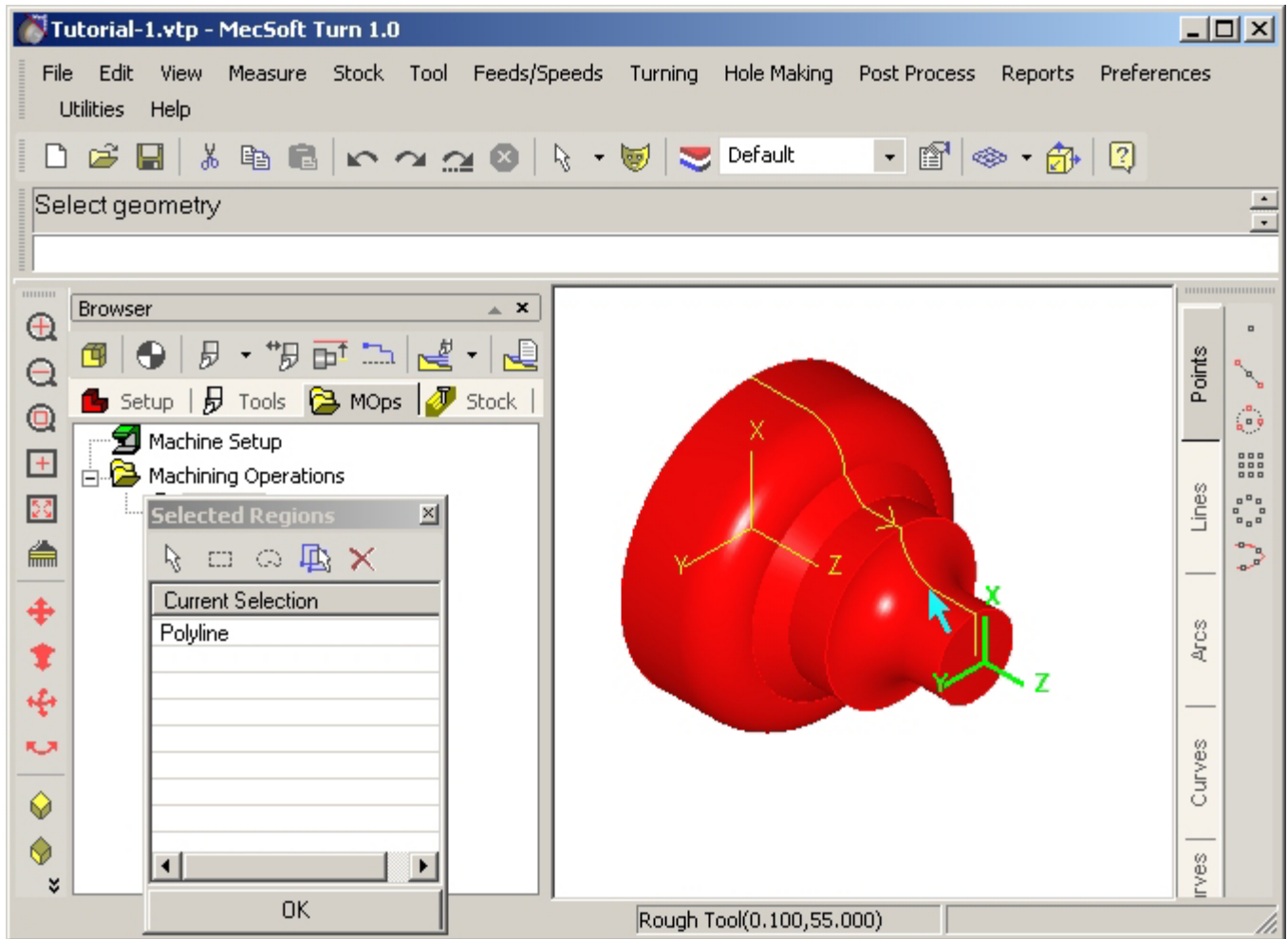


**To create the Outer Diameter Roughing toolpath:**

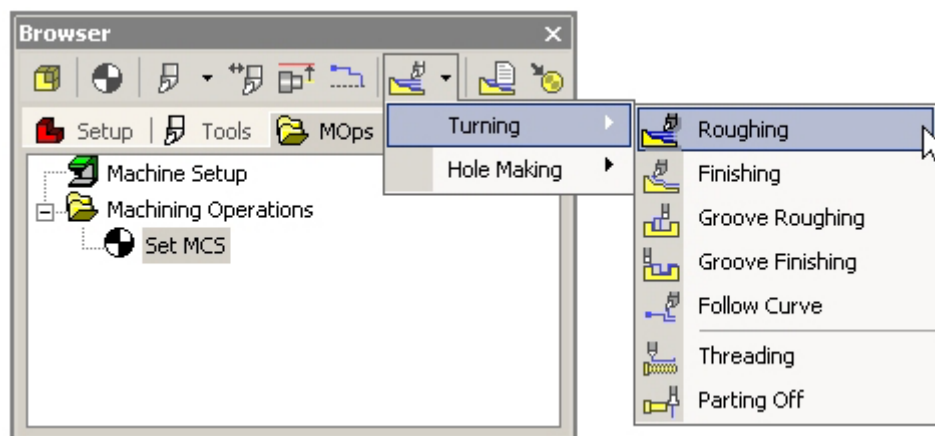
1. Once you have the tool selected and feeds and speeds set, you now have to select the region (2D profile of the geometry).
2. Go to **Select Regions** and use single select to select the 2D profile



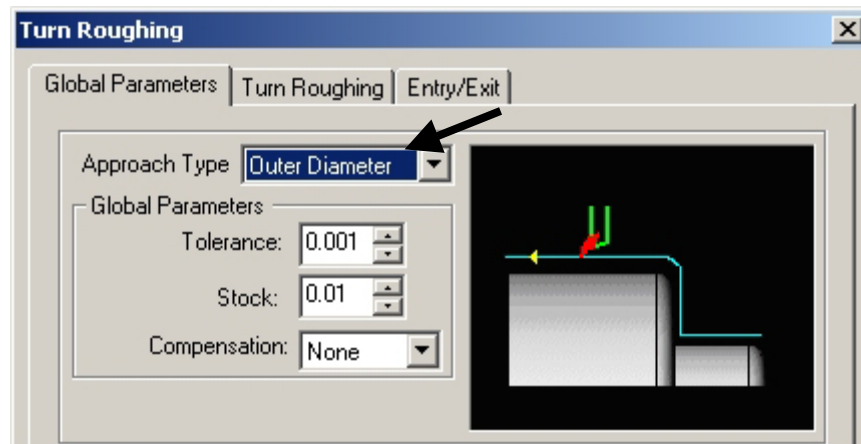
Getting Started with VisualTurn



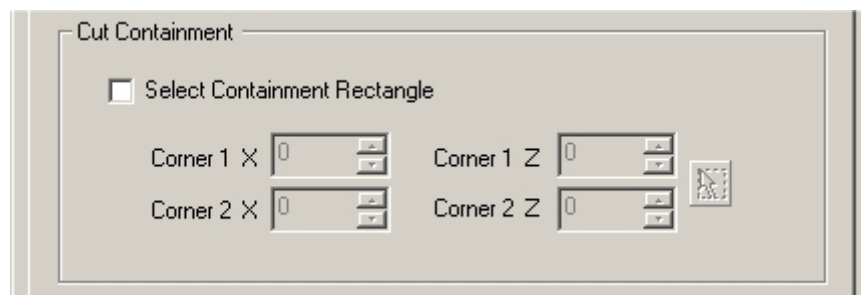
3. Click on the curve/polyline. This adds to the Selections Regions dialog. Click OK to complete selection.
4. Select **Turning / Roughing**, or click the **Machining** icon on the **Mops** tab of the Browser, and then select **Turning / Roughing**.



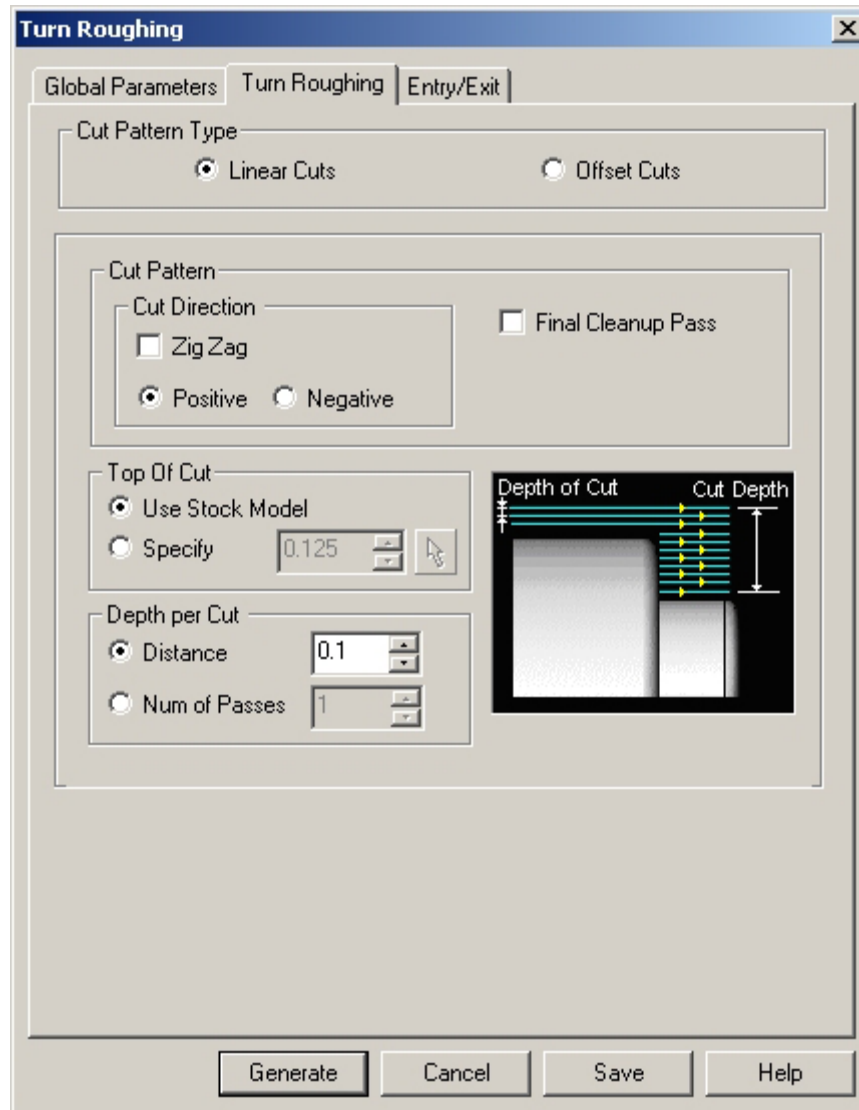
5. The **Roughing** window opens, in which you can set parameters for the toolpath.
6. In the **Global Parameters** tab, set the Approach type to **Outer Diameter**, the **Stock** value is set to 0.01 inches. This value defines the thickness of the layer that will remain on the part after the tool-path is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.



7. **Select Containment Rectangle** lets you create containment region if you wish to restrict the toolpath to a certain area only. To accomplish this Check **Select Containment Rectangle** button to enable the pointer that allows you to pick the containment rectangle. Click on the pointer to specify the region, defined by a rectangle. In this operation we will not be using a containment Rectangle.



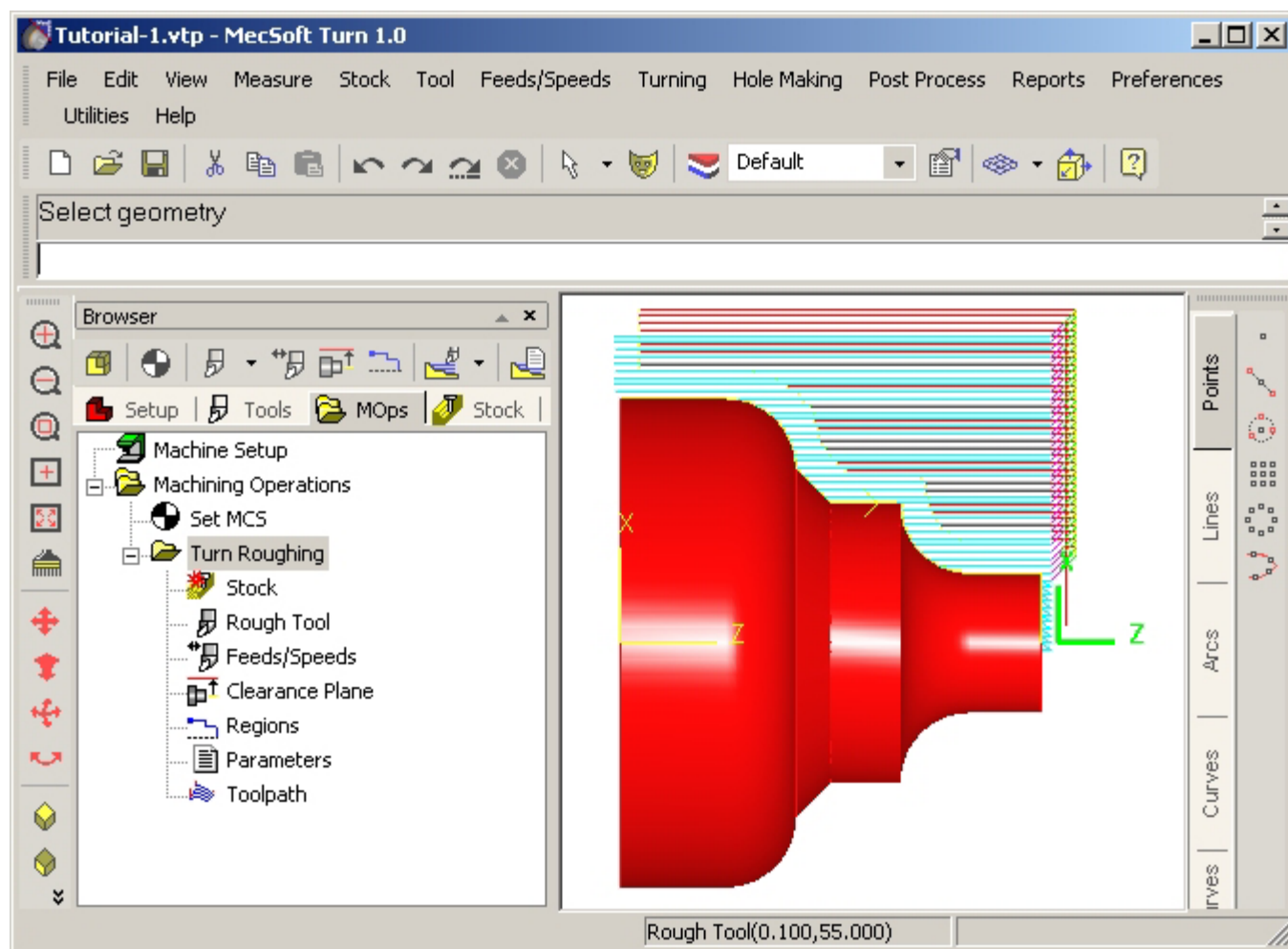
8. In the **Roughing Parameters** tab, set the **Cut Pattern** to **Linear**, uncheck **Final Cleanup Pass** and set **Depth per Cut** to 0.1.



9. Leave the remaining parameters in the other tabs as they are, and click **Generate**, located at the bottom of the window.

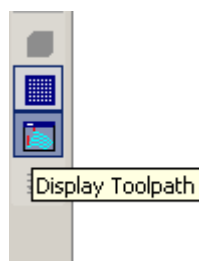
The window will disappear and an hourglass cursor will appear on the screen. When the computation is complete, the roughing toolpath will appear.

Note: See reference section for help on setting up Entry/Exit parameters

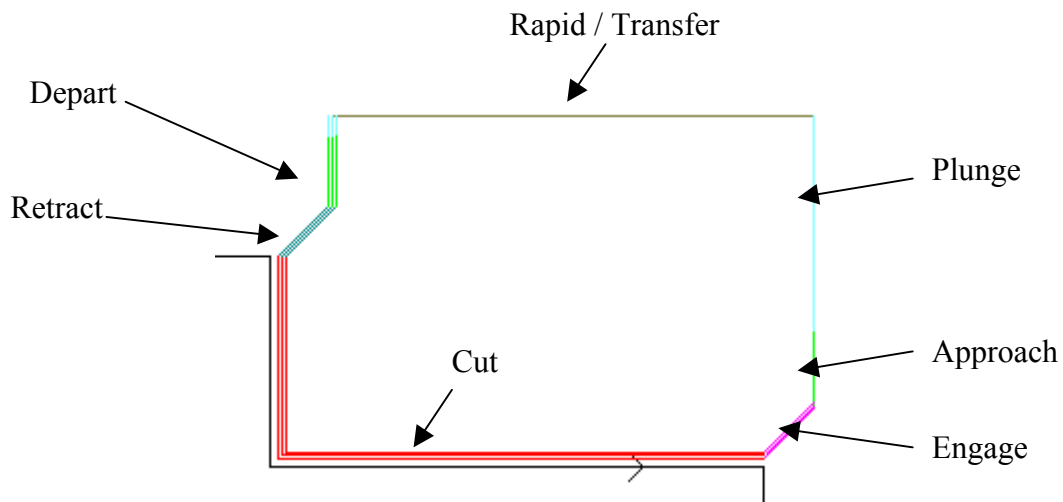


Note: You can control the toolpath colors by selecting **Preferences / Color Preferences**.

If the toolpath is not displayed, make sure **Hide Toolpath** is not selected.



Look in the **Mops** tab of the Browser, where you can see the toolpath you just created. Turn off the Default and Regions layers, so that only the toolpath is visible. You can see the various different types of motions. These are color-coded according to the table in **Preferences / Color Preferences** – check these colors if there are motions you cannot see.



Approach motions extend from the clearance plane down into the material.

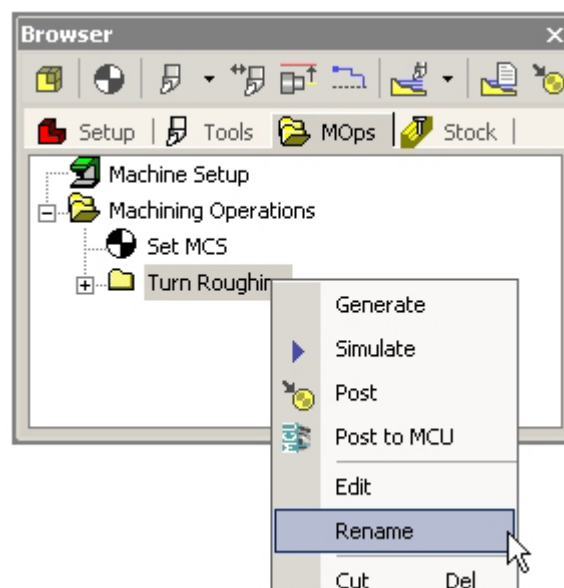
Cut motions represent actual cutting of material.

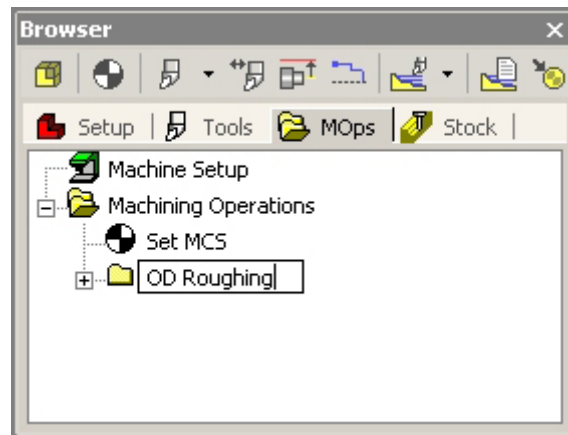
Depart motions extend from the material up to the clearance plane.

Rapid motions are along the clearance plane. They are fast because there is no danger of collision with material; the clearance plane is set a safe distance above the stock.

Retract motions come before **Depart** motions, allowing the tool to exit the cut material safely. Likewise, **Engage** motions come after **Approach** motions, allowing the tool to engage into the cut material safely.

Right-Click on the Machine Operation name, i.e. **Turn Roughing** to edit it. Change it to **OD Roughing**.





Note: In order to rename an operation single select on a Machining operation name and use the right mouse click to rename

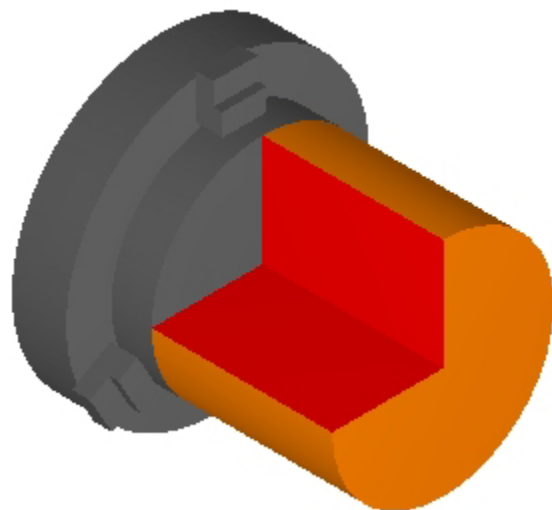
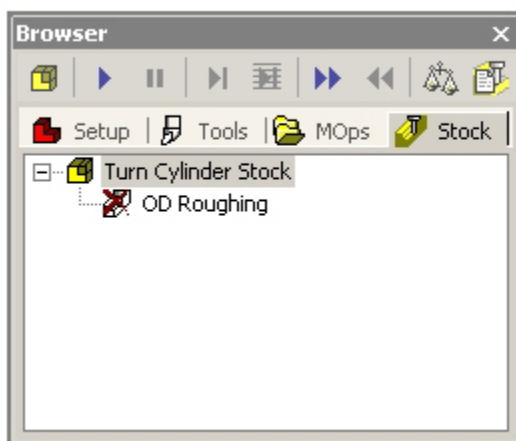
Simulating the Outer Diameter Roughing Toolpath

Now that the first toolpath has been created, you can simulate it.



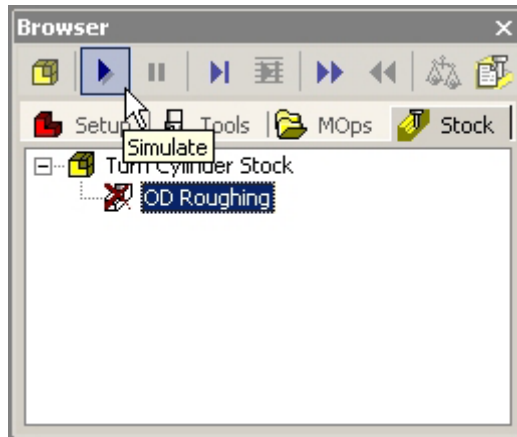
To simulate the toolpath:

1. To see how the stock looks after this toolpath, switch to the **Stock** tab. The cylinder stock box is displayed. The toolpath name is displayed with a red X, indicating that the simulation has not been run. (Click on Turn Cylinder Stock to view the stock)

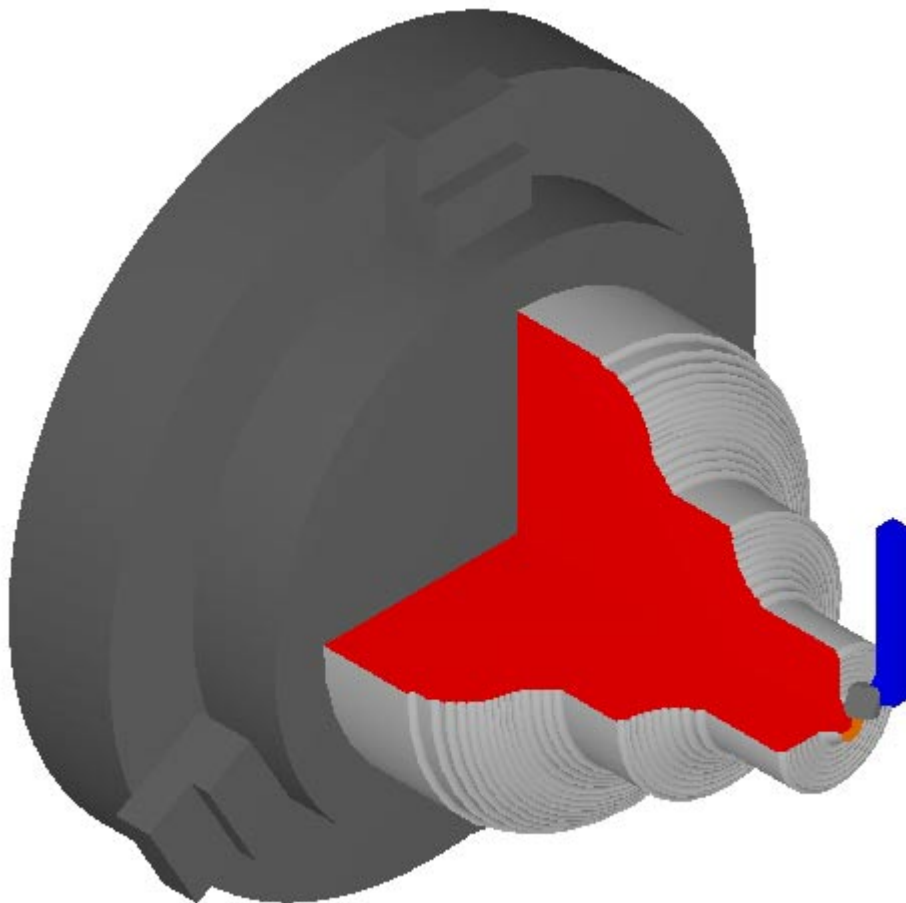


Getting Started with VisualTurn

2. Highlight the **OD Roughing** toolpath and click **Simulate**.



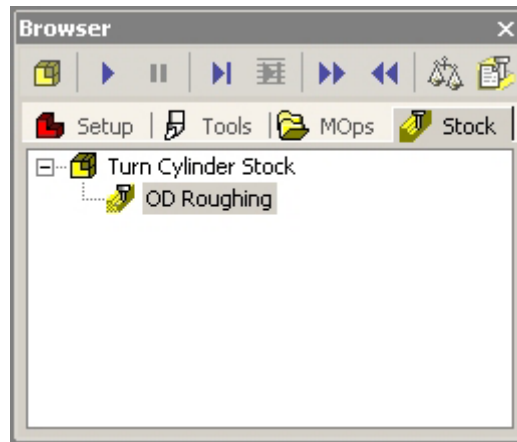
Once the simulation is complete, the cut stock model will be displayed. This cut model can be used as input stock geometry for simulating the toolpath of subsequent machining operations.



Note: You can also try clicking **Step** simulation to view a set number of tool motions at one time.

Tips: You can change the color of the stock and cut stock by selecting **Preferences / Color Preferences** and clicking the color box for **Cut Stock Color**.

The toolpath now has a “simulation complete” icon next to its name in the **Stock** tab.



Creating the Outer Diameter Finishing Toolpath

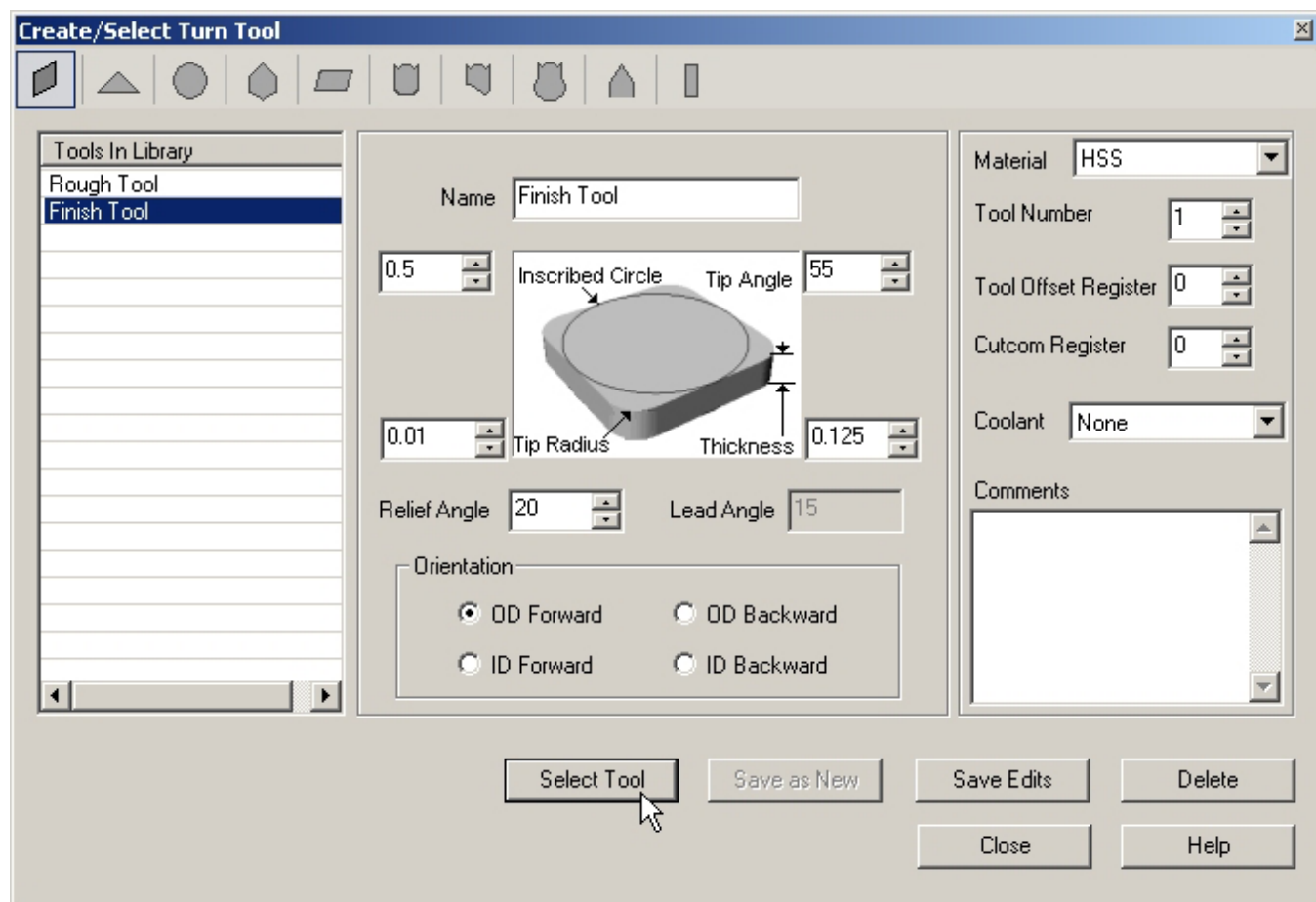
Once the roughing toolpath is generated, a finish toolpath can be created to remove the steps left by the roughing process, and to bring the stock equal to the part. In this exercise will use the **Finishing** method.



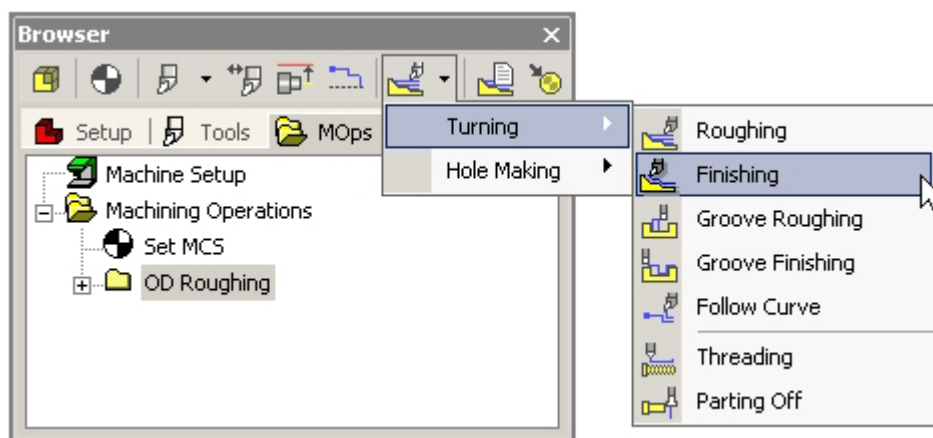
To create the Outer Diameter Finishing toolpath:

1. Return to the **Mops** tab.
2. Activate the **Finish Tool** by clicking on the create Turn/Drill tool from the Mops tab.

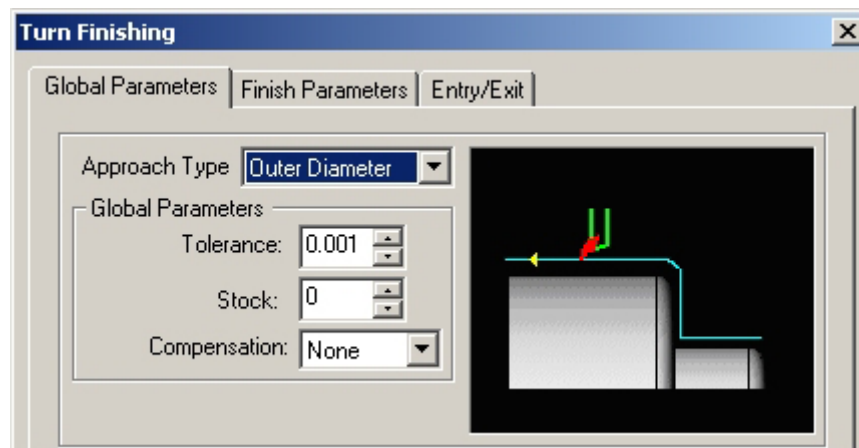
Getting Started with VisualTurn



3. Select **Turning / Finishing**.



4. Set the Approach type to **Outer Diameter** and **Stock** to 0

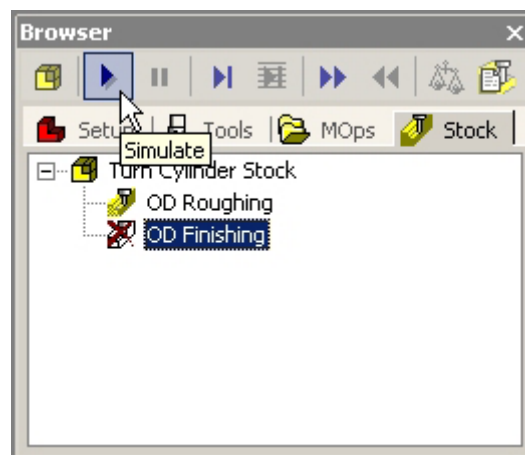


5. With the default set of values in the **Global and Finish Parameters** click **Generate**
6. Rename this operation **OD Finishing**.

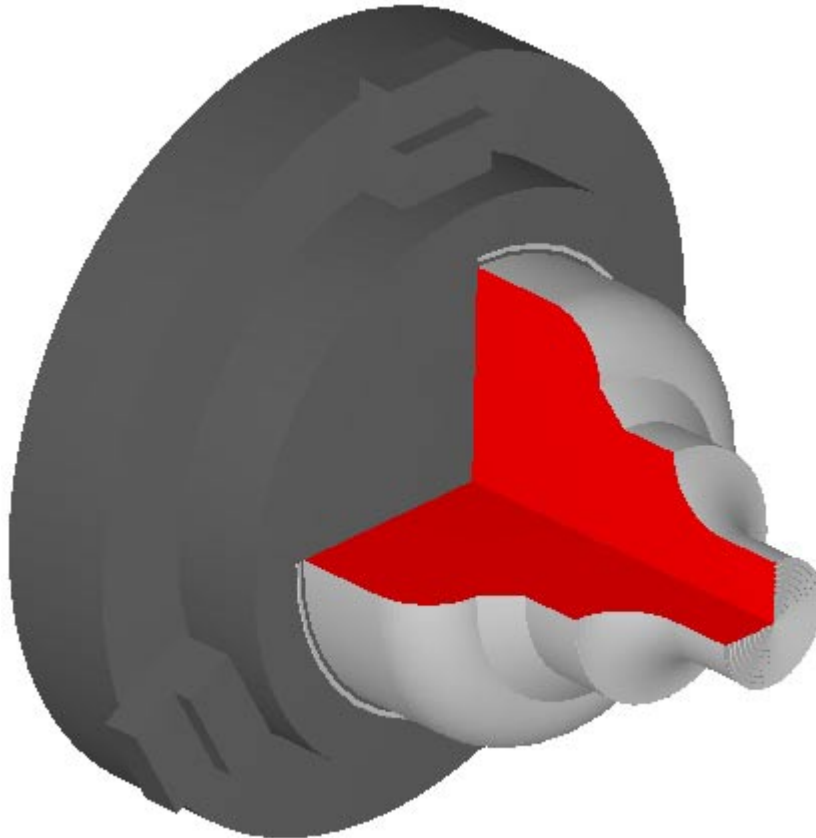


To simulate the outer diameter finishing toolpath:

1. Switch to the **Stock** tab. Select **OD Finishing**.



2. Click to **Simulate**.



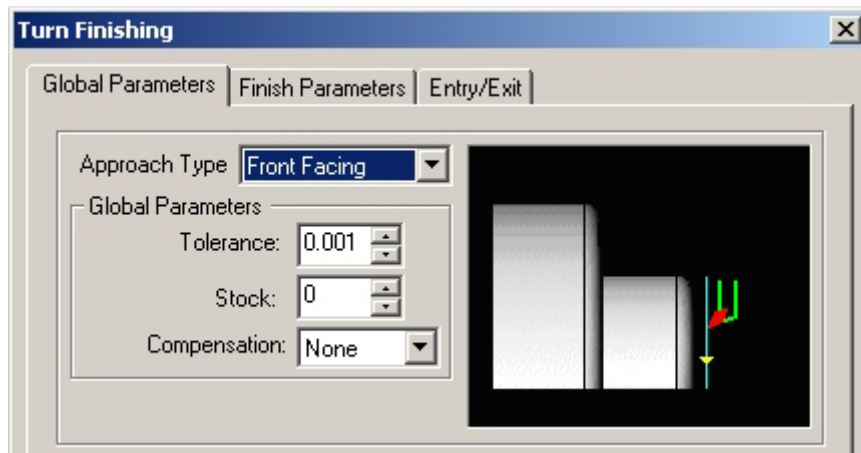
Creating Face Finish Toolpath

A finish toolpath can be created to remove the steps left by the roughing process, and to bring the stock equal to the part. In this exercise will use the **Finishing** method.

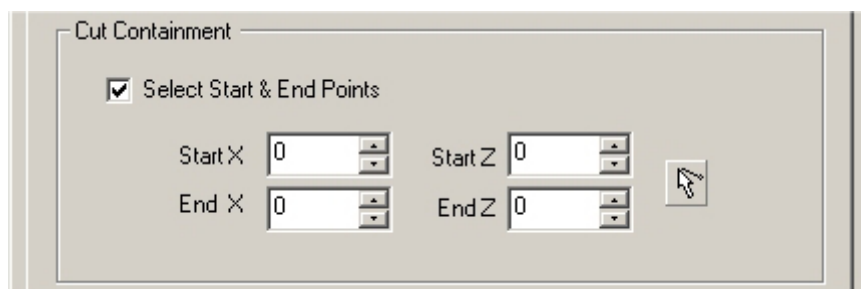


To Create the Face Finishing Toolpath

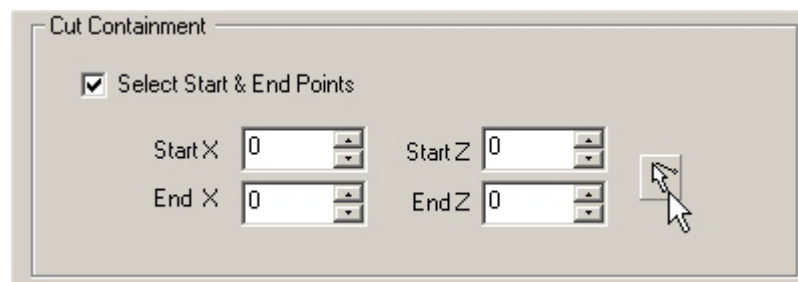
1. Return to the **Mops** Tab
2. With the **Finish Tool** selected under Machining Methods Select **Turning / Finishing**
3. Set the Approach Type to **Front Facing** and **Stock** to **0**.



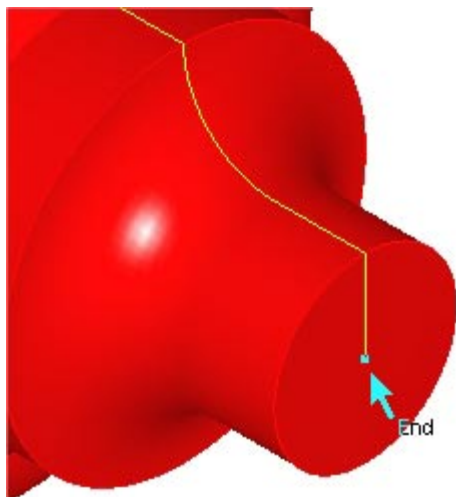
4. As we have created finishing operation on the outer diameter of the geometry and only the front face remains we will specify a containment region by setting Cut Containment Check at Start and End Points



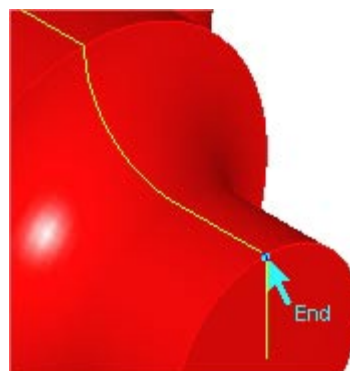
5. With the Mouse select tool for Start and End select the start and End points as shown below. Make sure the End Snap is turned on.
6. Clicking the Mouse Select minimizes the Turn Finishing dialog



Getting Started with VisualTurn



Start Point Selection



End Point Selection

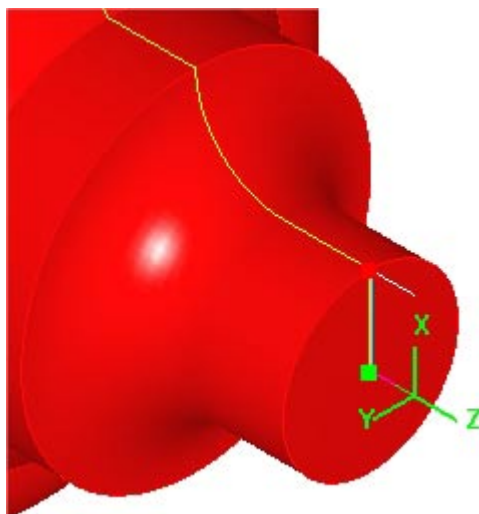
7. Once you have selected the Start and End points, the cut containment should have the coordinate values for Start and End

Cut Containment

☒ Select Start & End Points

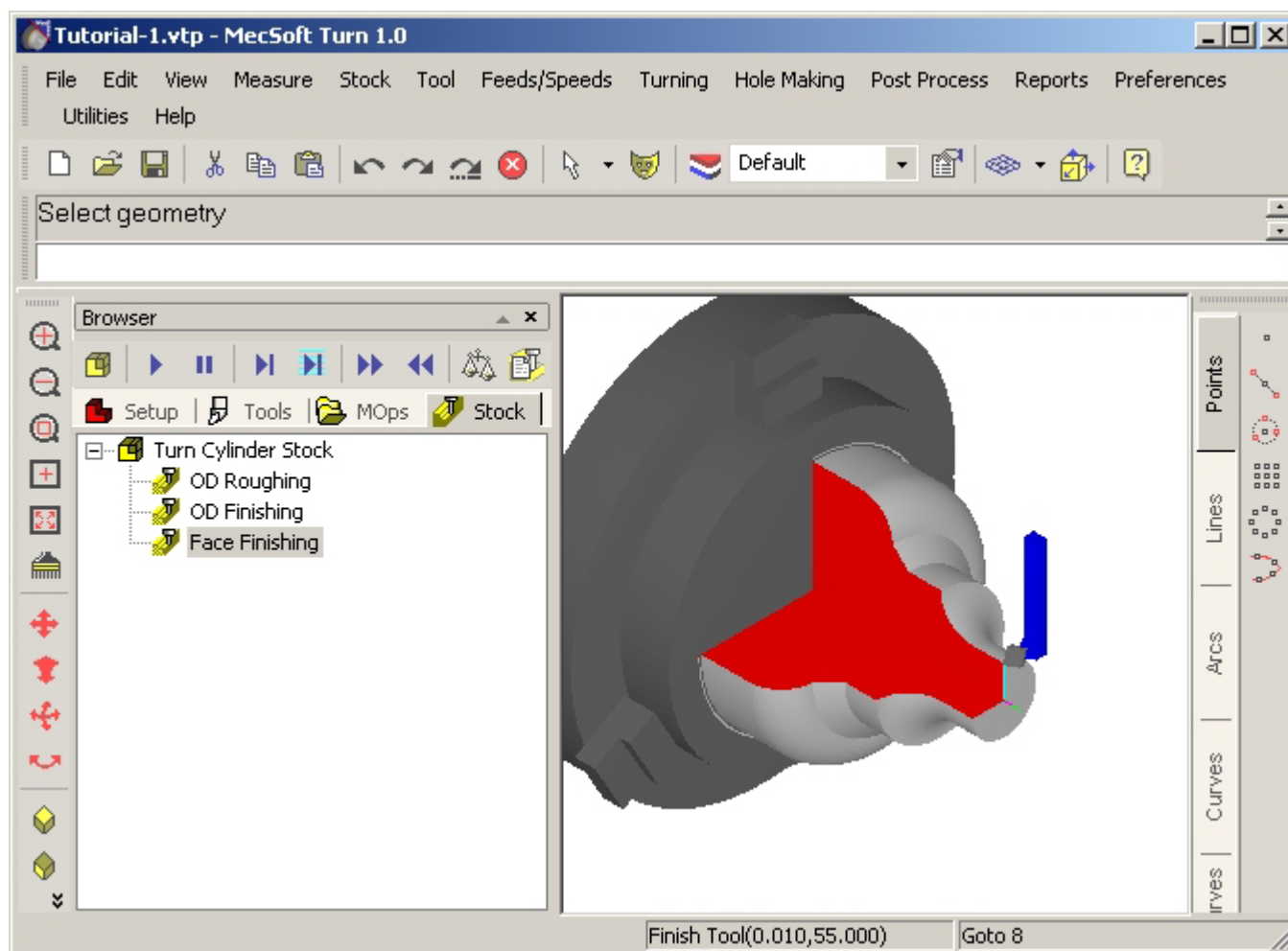
Start X	0	Start Z	6
End X	1.00100	End Z	6

8. With the other parameters set to default we will now click generate to create the Front Facing toolpath.



9. Rename the Operation to Face Finishing

10. Switch to the **Stock** tab and select Face Finishing to Simulate.



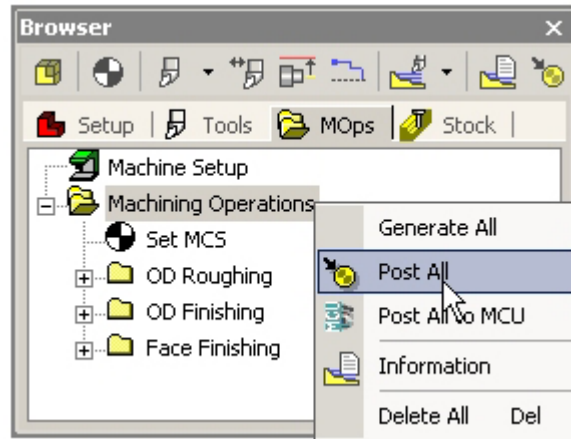
Look under Simulation settings  to change the simulation speed and simulation accuracy.



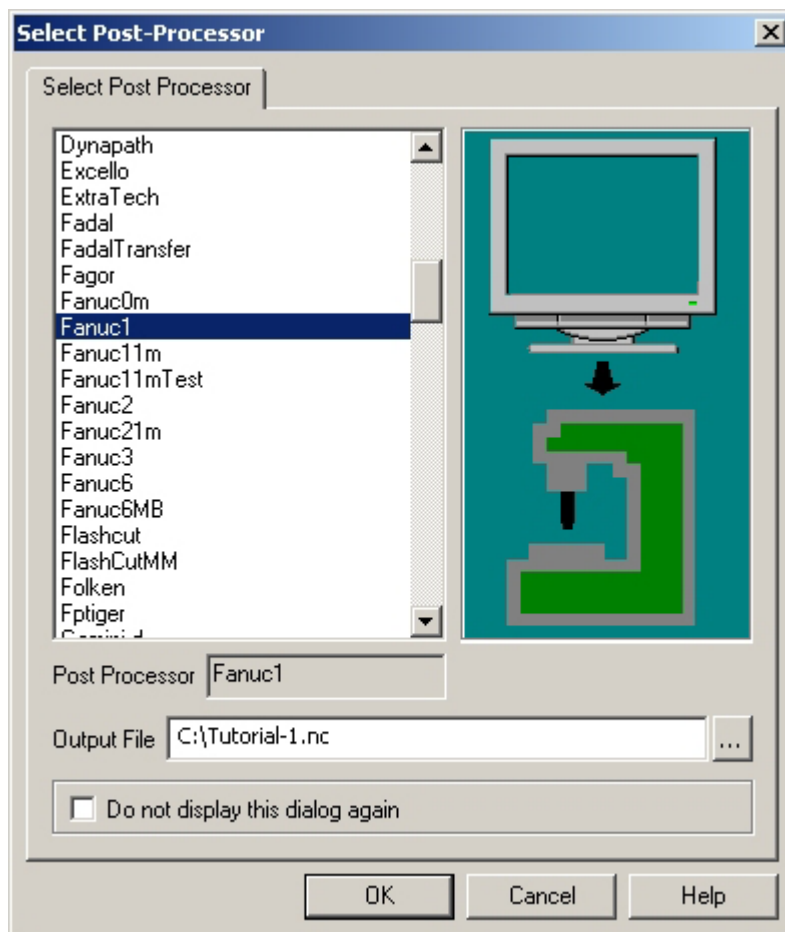
To create the post-processed output:

In this exercise we will post-process all of the toolpaths at once.

1. In the **Mops** tab, right-click on the root folder (**Machining Operations**) to open the popup menu. Select **Post All**.



2. Browse to the desired output directory and assign a file name for the output. The default extension is *.nc. Then double-click on the post-processor you want to use (such as **Fanuc 1**).



3. When complete, the post output file will open in the default text editor (Notepad by default). This file contains all the G-code for your toolpaths.

***Note:** You can post individual toolpaths by right clicking on their name in the **Mops** tab and selecting **Post**. The **Post-Process** icon on the **Mops** tab can also be used. To post multiple toolpaths, select each toolpath while keeping Ctrl pressed, right-click, and select **Post All**.*

End of Tutorial 1!

Tutorial 2: ID Roughing, Finishing and Drilling

In this tutorial, you learn to create Inner Diameter (ID) roughing, finishing and drilling toolpaths

The stepped instructions are accompanied by explanatory and introductory text. Reading this text will help you understand the tutorial methodology and provide information about additional options available. However, if you prefer to work straight through the steps without any additional reading, look for the following symbol:



Don't forget to save your work periodically! You may want to save the file under a different name so that the original file will be preserved.

This exercise will help you understand and use the drilling module in VisualTurn. Under 2-axis turning, holes can be drilled only along the Z-axis in the center of the part.

The following types of drilling operations are available:

1. **Drill:** Standard, Deep, Break chip, Counter Sink
2. **Tap:** Clockwise, Counter Clockwise
3. **Bore:** Drag, No Drag, Manual
4. **Reverse Bore**

Creating the Axial hole

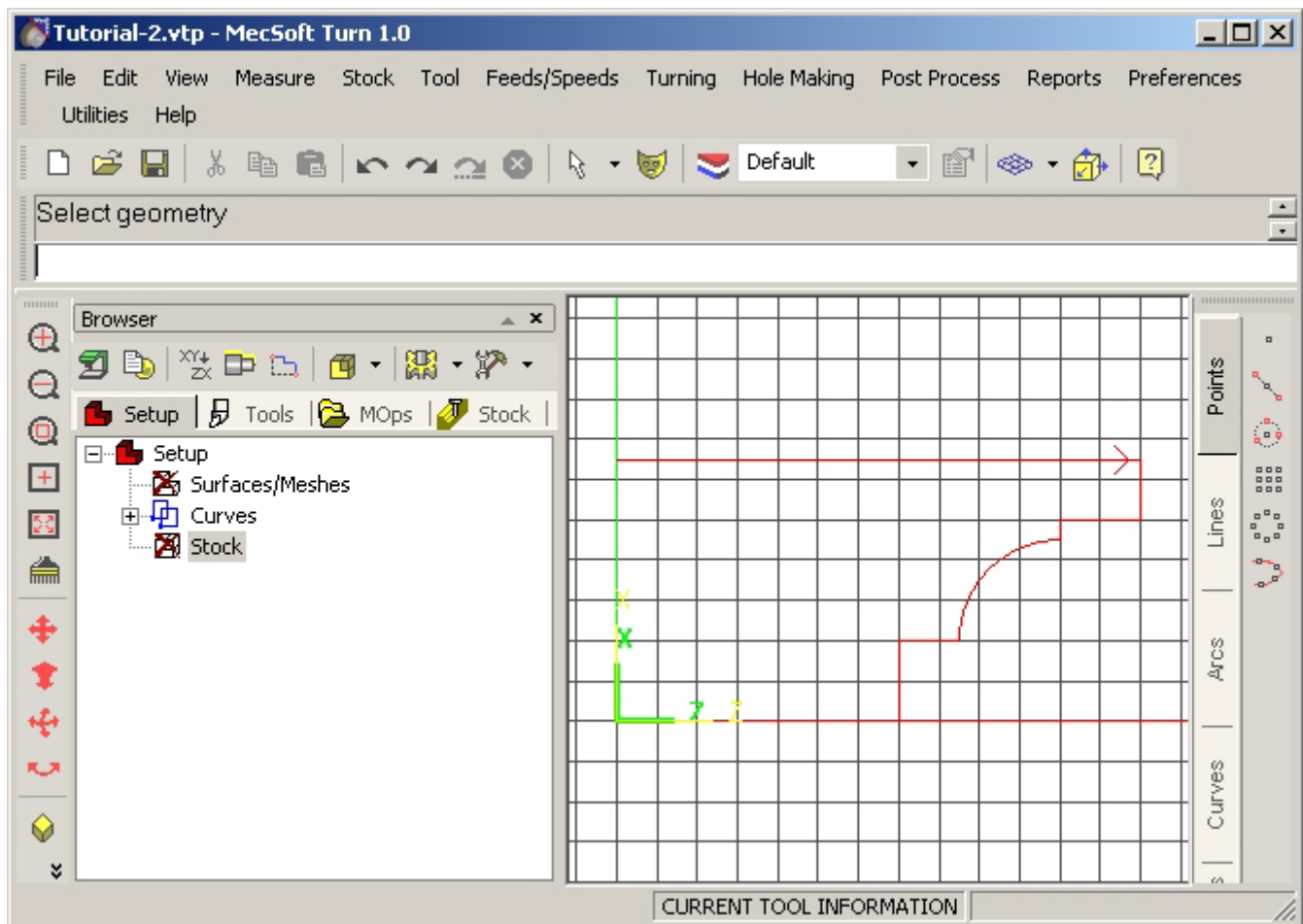
The first operation we will create is to make an axial hole in the center of the part so that we can employ ID tools to create the ID shape of the part.



To create a drilling operation

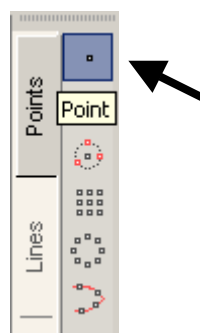
1. Select **File / Open**, or click the **Open Part File** icon from the **Standard** bar. Select the **Tutorial-2.vtp** file from the **Tutorials**.

Note: To turn on Grid Display click on  Display Grid from the View bar



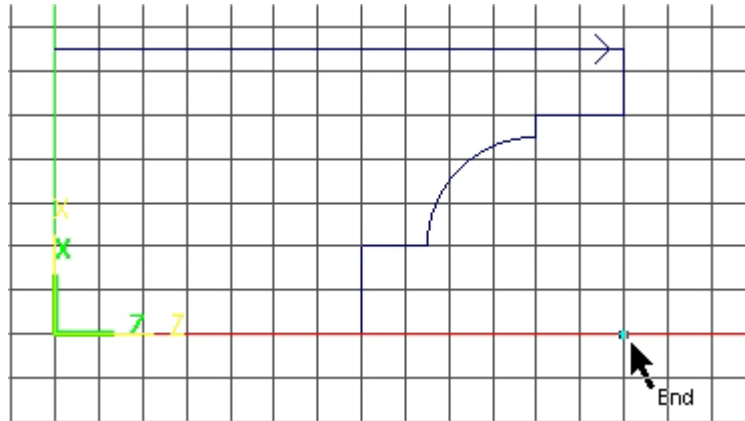
2. Use the point select tool from the geometry bar to create a point at $X=0$ and $Z=13$

Point coordinates can also be specified using the command bar as 0,0,13 (X, Y, Z)

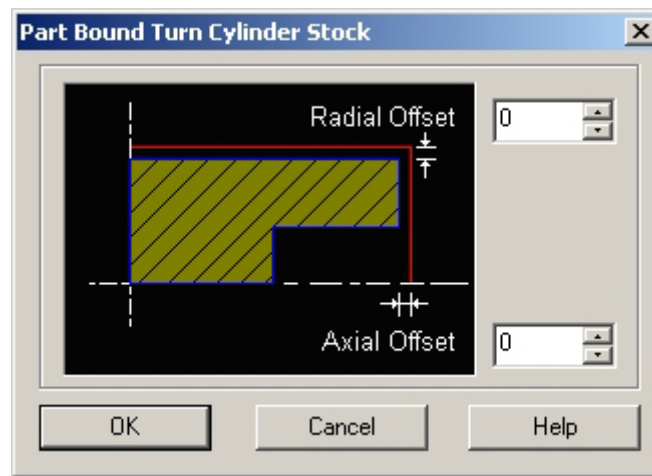


3. The point created is as shown below

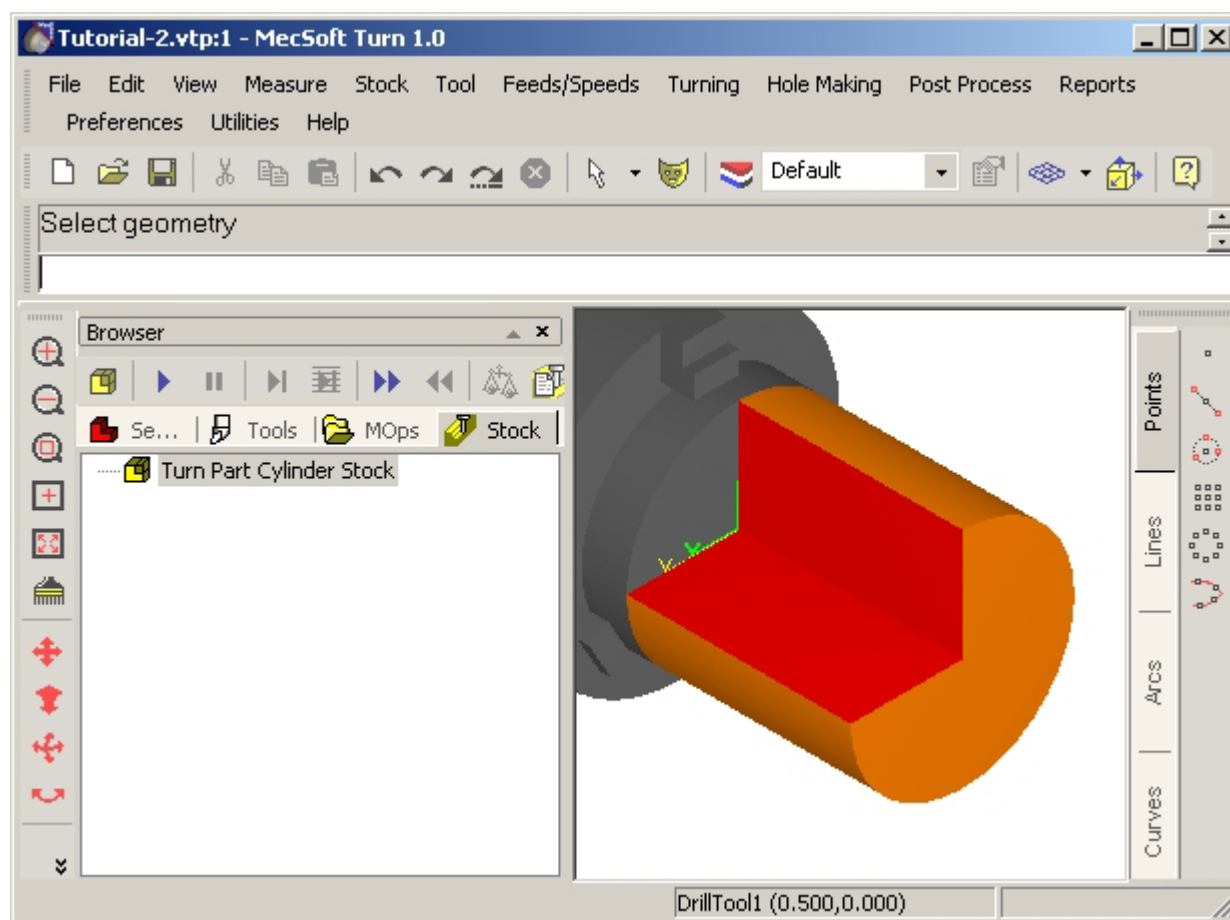
Getting Started with VisualTurn



- Click **Create/Load Stock** from the **Setup** tab of the Browser and select **Part Cylinder Stock**.

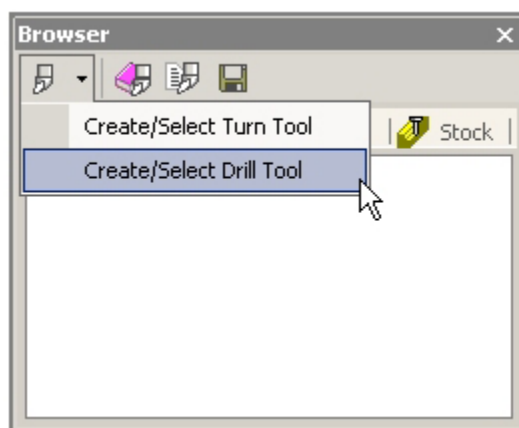


- Set **Axial** and **Radial** offset to **0**.
- The stock model is created. To display the stock, click the **Stock** tab of the Browser. The stock is displayed as a cylinder positioned at the reference point of the lathe machine. Its color can be set in the **Color Preferences**.

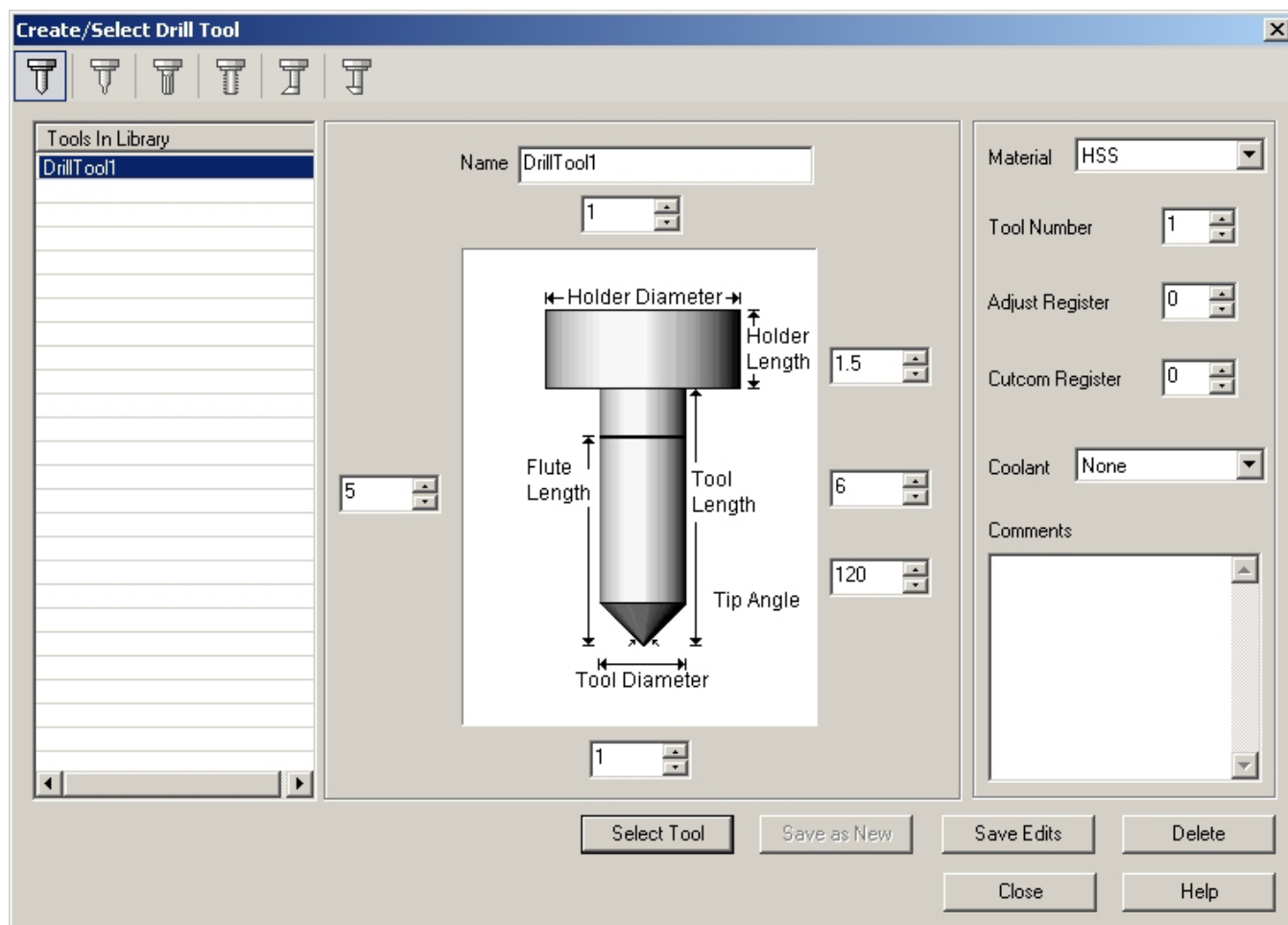


To create a Drill Tool

1. Switch to the tools tab and click on create/select drill tool

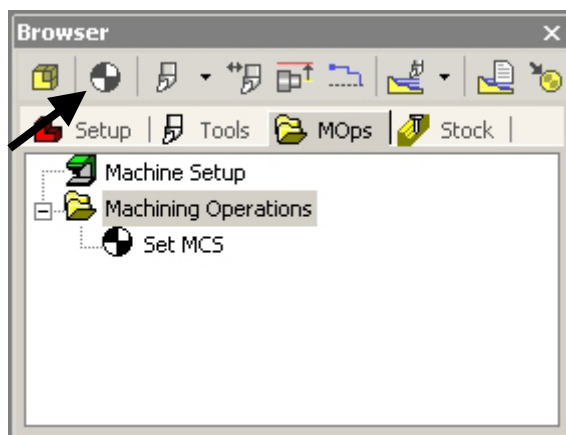


2. Create a **standard drill** and set the tool diameter to 1", Flute length to 5", Total length to 6". Leave the other parameters at default. Click Close to exit the dialog.

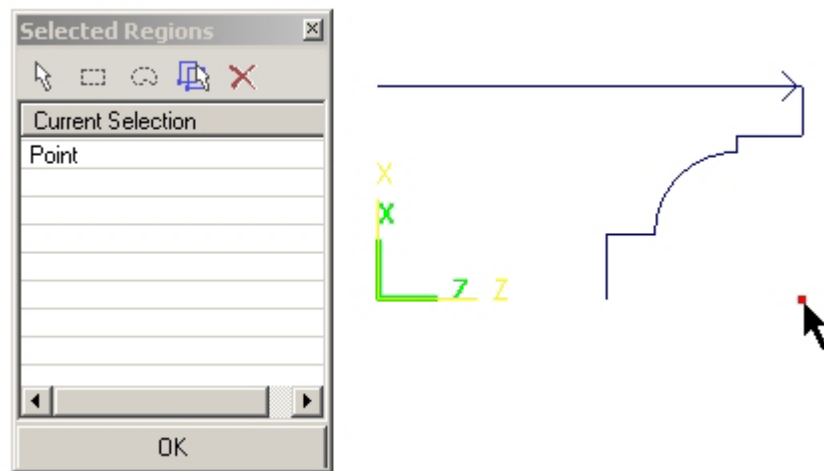


To create a drilling operation

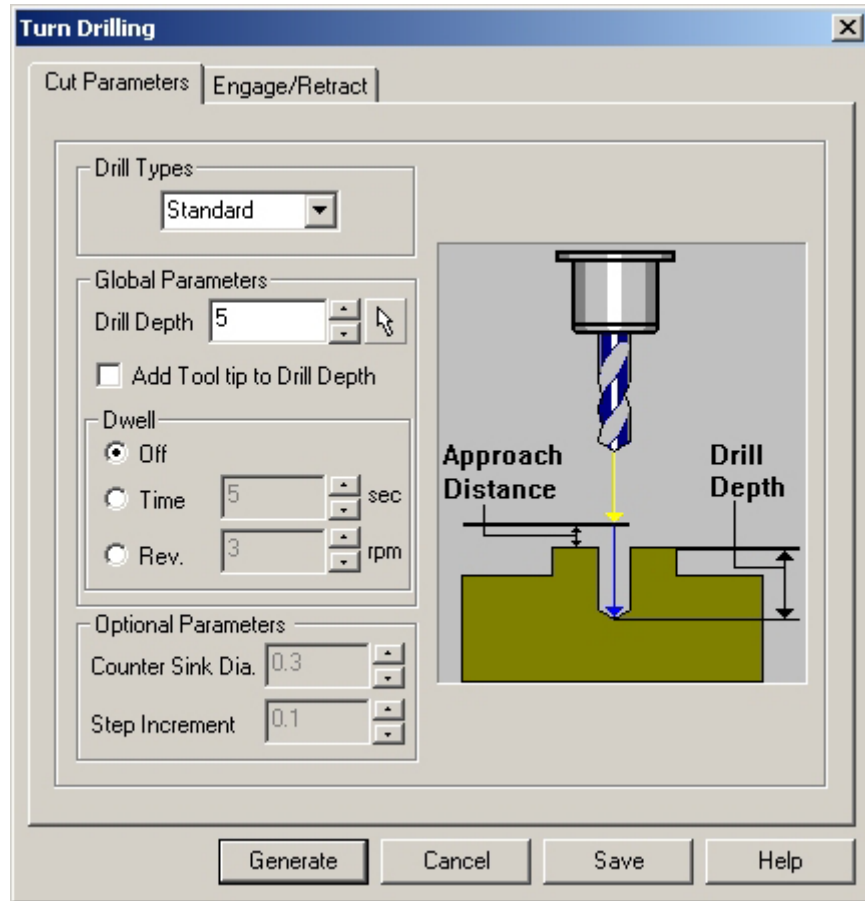
1. Switch to the Mops tab
2. Click on the **Set MCS** icon in the **Mops** tab of the Browser Bar or Double-Click on the Set MCS under Machining Operations



3. For this example, set the **MCS** origin to the **Stock Box** and Zero Face to **Right Most** face of the part. Click **OK**
4. Click on **Create/Select Drill tool** and select the **Drill tool**.
5. We will leave the feeds and speeds with the default settings. Clearance Plane is set to automatic by default.
6. From Select regions use single select to select the point that was created for the drilling operation.



7. Select **Hole Making** under machining methods and choose **Drilling**
8. Set the drill type to standard and drill depth to **5"** and leave the rest with default set of values and click Generate. The toolpath is now generated.

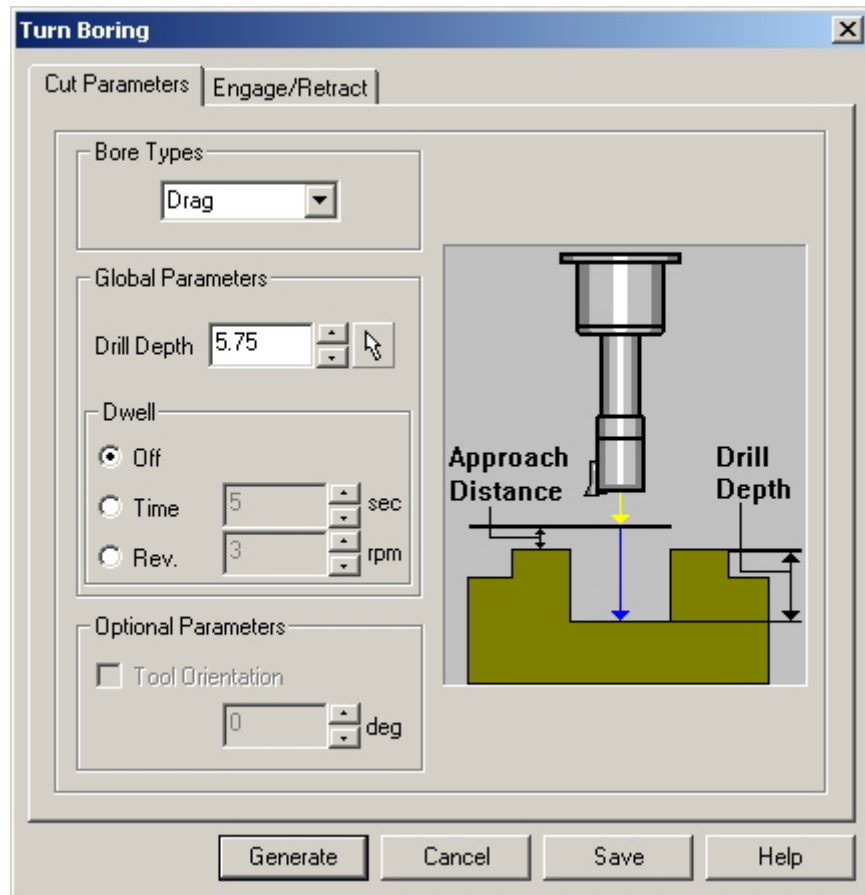


9. Switch to the stock tab, select the **Standard Drill** operation and click  to simulate.



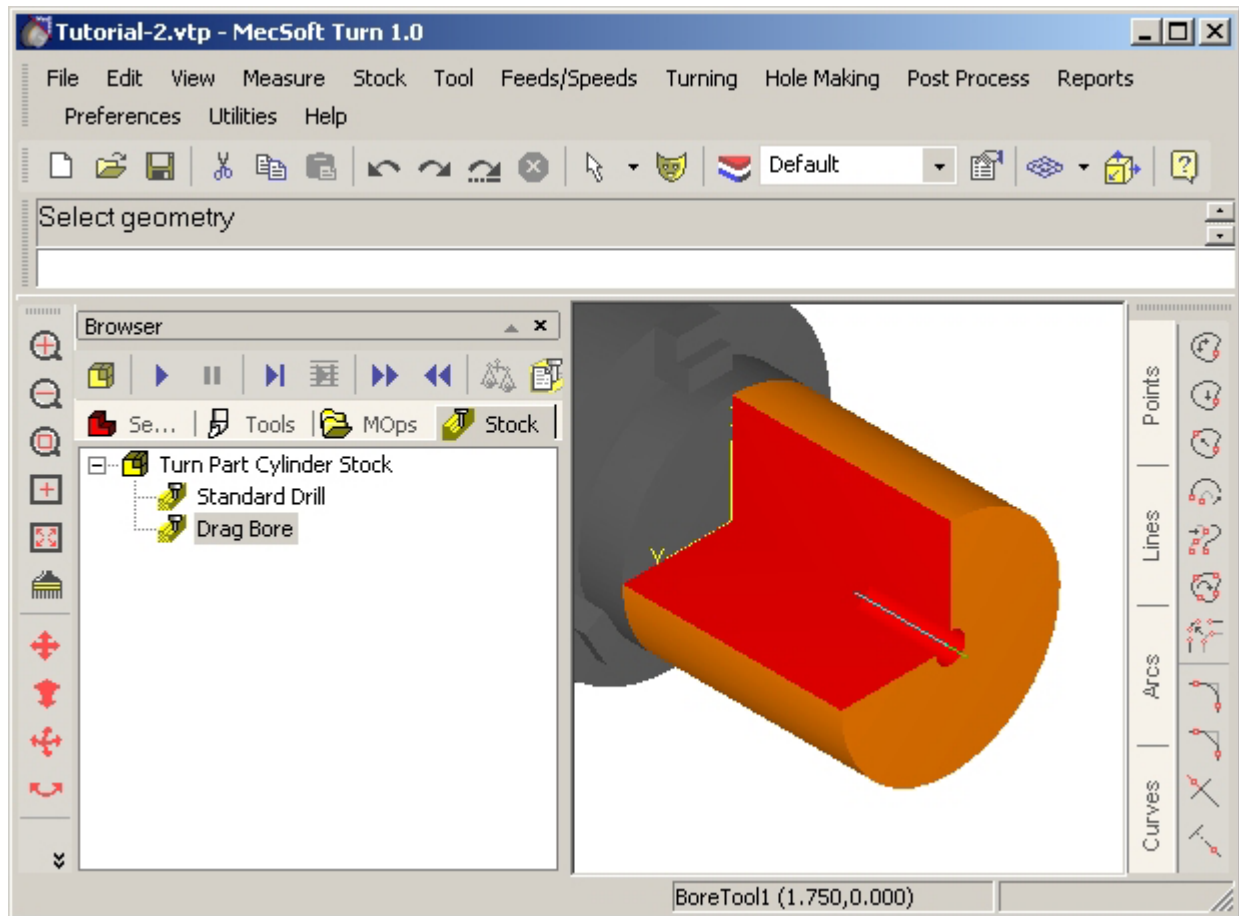
To create a boring operation

1. Switch to the Mops tab
2. Click on **Create/Select Drill tool** and create a **bore tool** with the following parameters:
Diameter of 1.75", Flute length 5.5", Tool length 6.5", Shank Dia 1.75", Holder Diameter 2"
3. From Select regions use **Single select** the select the point that was used for the drilling operation.
4. Select **Hole Making** under machining methods and choose **Boring**
5. Set the bore type to drag and drill depth to **5.75"** and leave the rest with default set of values and click generate. The toolpath is now generated.



6. Switch to the stock tab, select the **Drag Bore** operation and click  to Simulate.

Getting Started with VisualTurn



(The simulation settings are set to 3-quarter view)

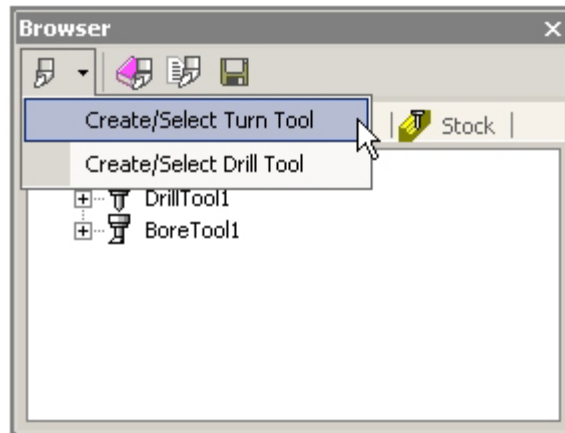
Creating the Inner Diameter Roughing Toolpath

Once the boring toolpath is generated, a rough toolpath can be created to remove more material to bring the shape closer to net shape.

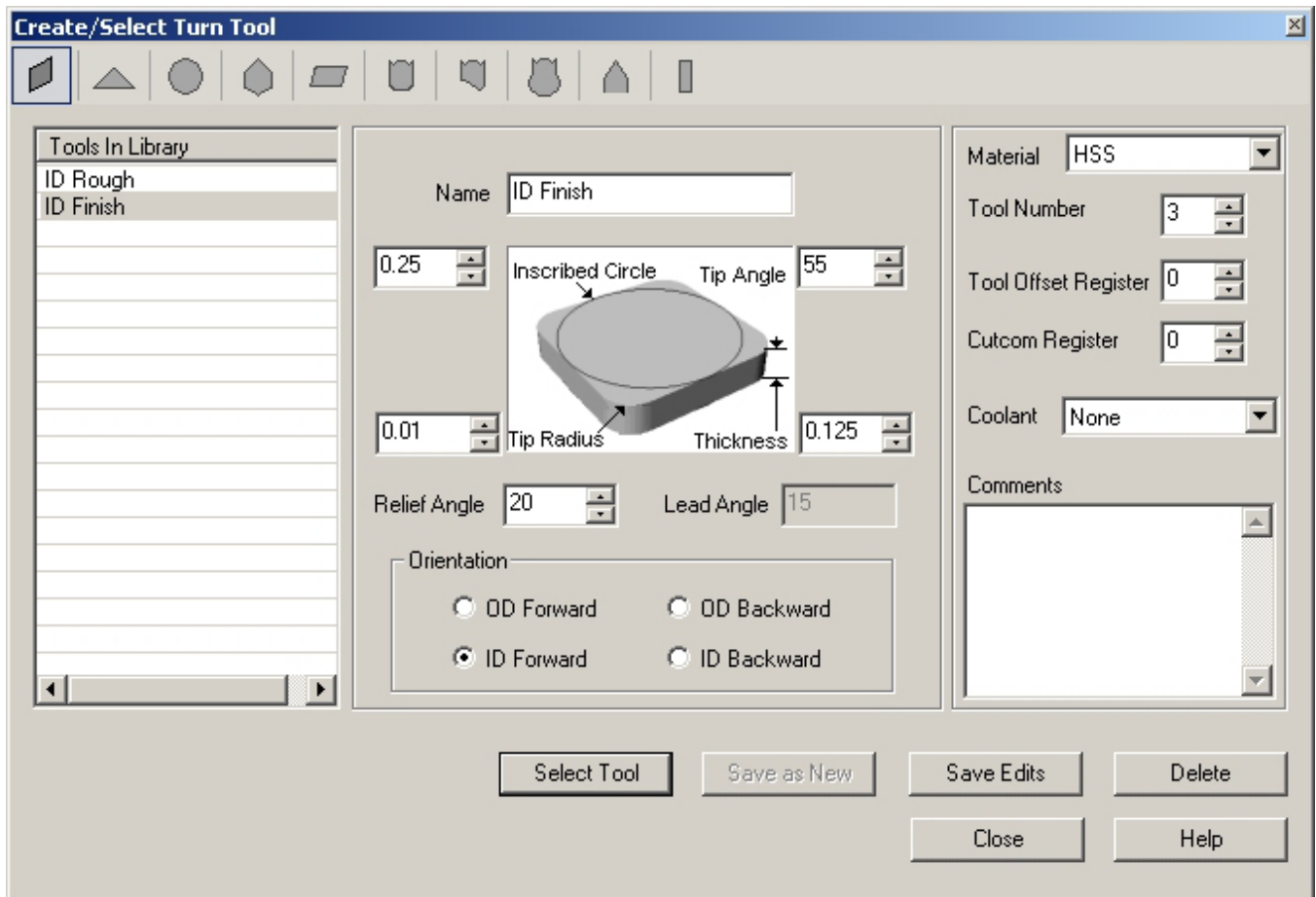


Steps for Creating ID Roughing Toolpath

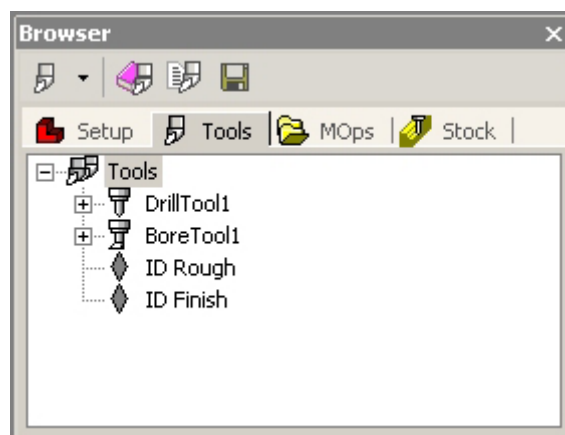
1. Select **Tool / Create/Select Turn Tool**, or click the **Create/Select Turn Tool** icon on the **Tools** tab of the Browser.



2. In the **Select/Create Turn Tool** window, click the **Diamond** tab. Change the name to **ID Rough Tool** and Tip Radius to 0.1", Inscribe Circle to 0.25". Set the Orientation to ID Forward and choose the default values for other parameters and click **Save as New**.
3. While still in the **Diamond insert** tab, create another tool by setting the tip radius to 0.01 inches as finishing is typically performed with a smaller radius tool.
4. Change the tool name to **ID Finish Tool**. Make sure that the Orientation is set to ID Forward
5. Click **Save as New**.



6. Click **Close** to close the window.
7. Now that all tools have been created, click the **Tools** tab in the Browser. All the tools are listed.



Note: You can double-click on any tool to open its definition window. This is an easy way to make changes, if needed.



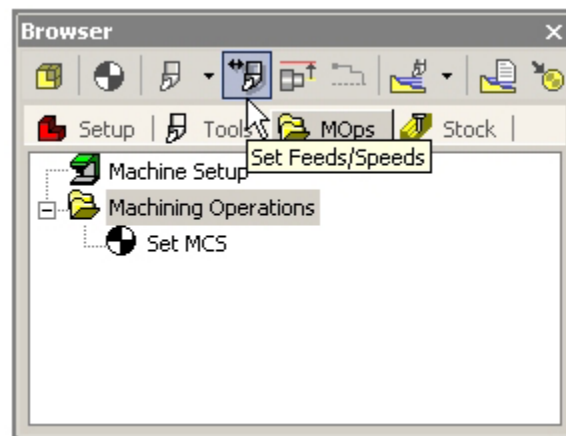
Selecting the Tool for ID Roughing Operation

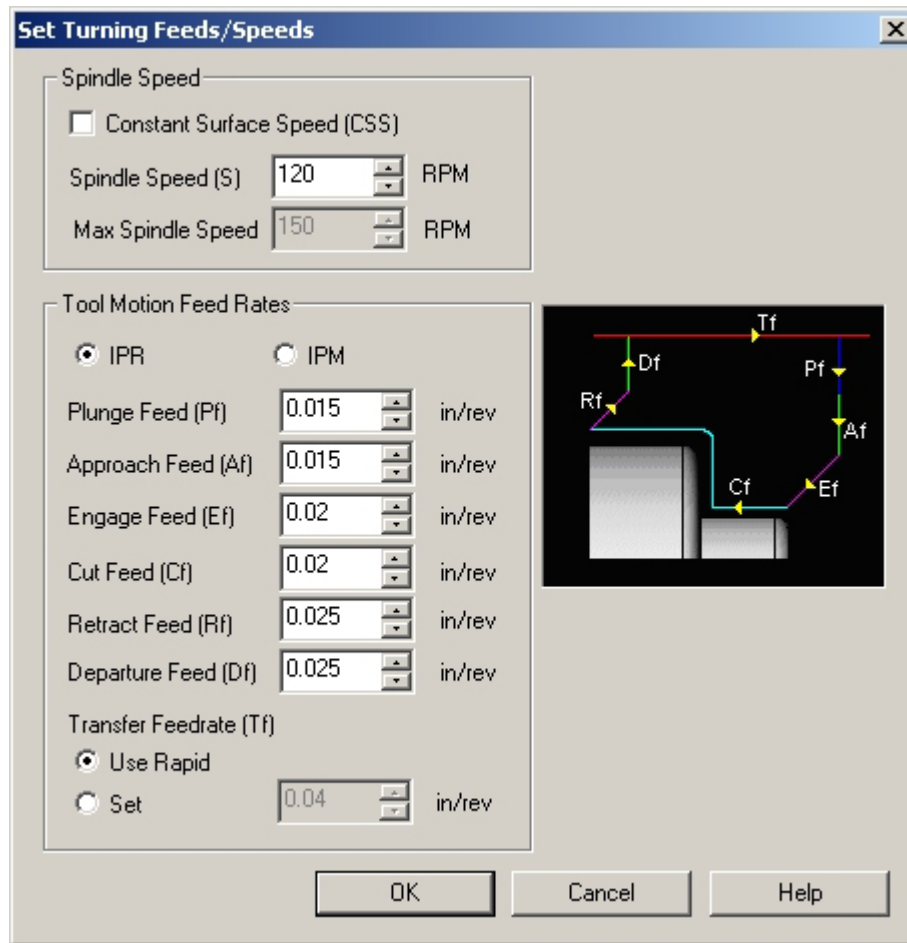
1. Under the Mops tab click on Create/Select Turn tool
2. From the tool select dialog pick the ID Rough tool and click select tool. This will make the Rough tool as the active tool and shows up in the status bar at the bottom of the screen



Setting Feeds and Speeds

You can set toolpath feeds and speeds and customize these settings for later use. To set the feeds and speeds click on Set Feeds/Speeds on the Mops tab. This launches the Feeds/Speeds dialog.



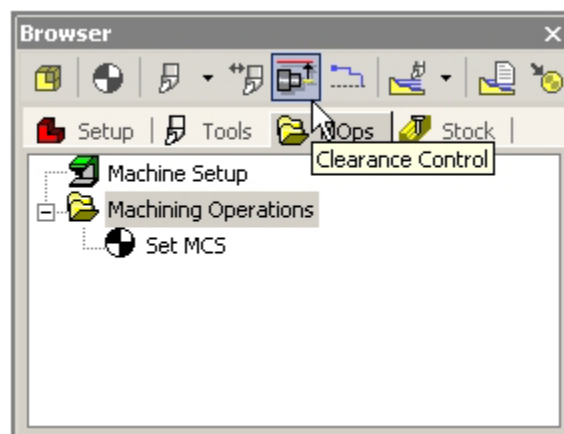


Feeds & Speeds: Considering Stock material as Aluminum and Tool material as HSS for the above example.

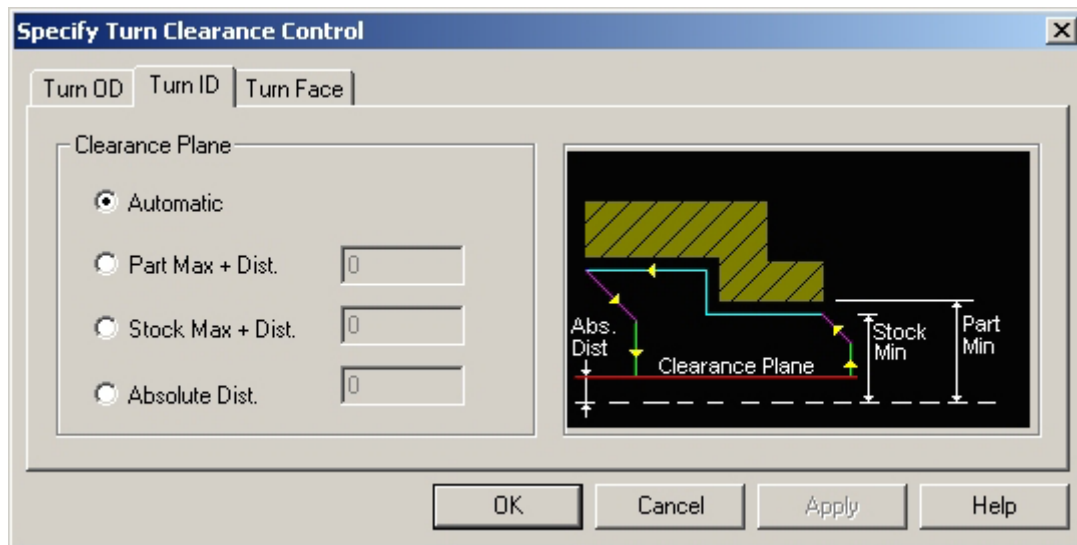


Clearance Plane

1. Clearance levels are set by clicking the **Clearance Control** button on the **Mops** tab.

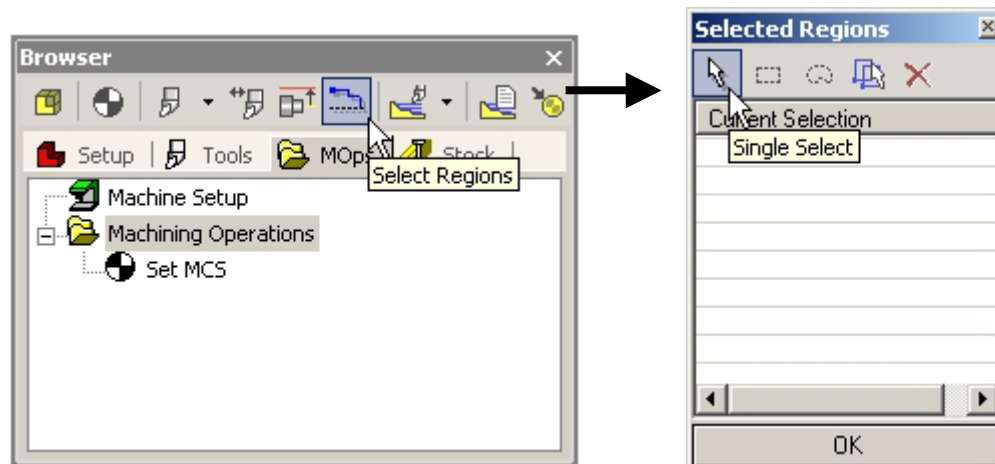


- By default the Clearance Distance is set too automatic.

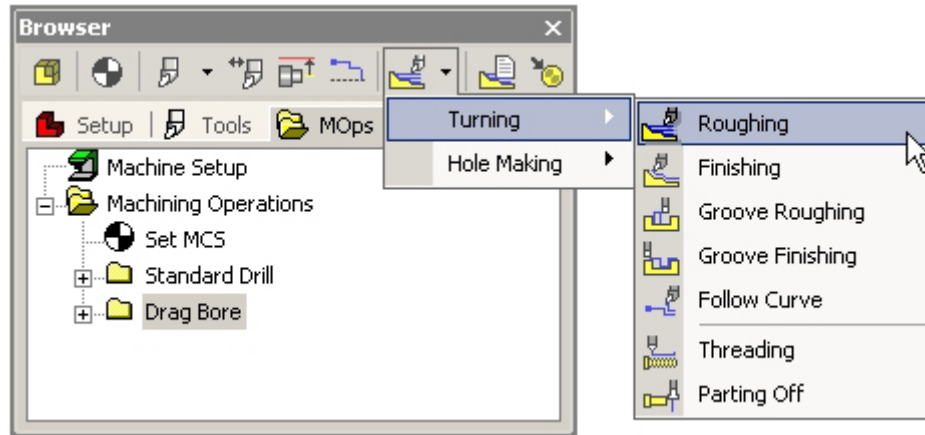


To create the Inner Diameter Roughing toolpath:

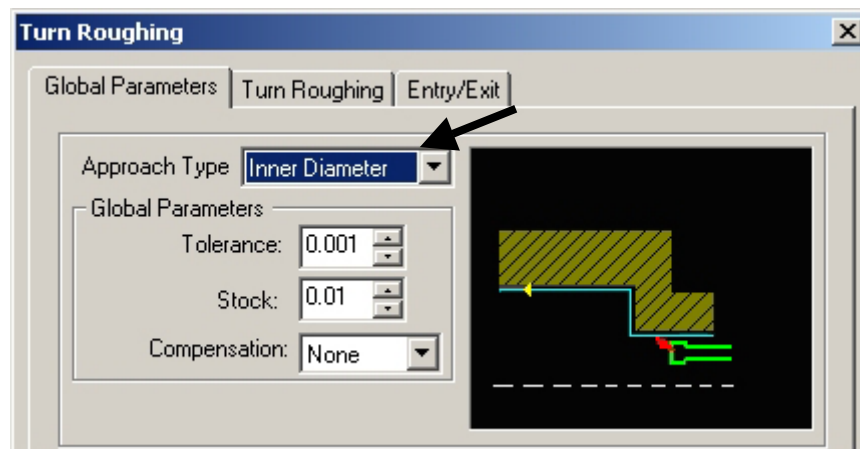
- Once you have the tool selected and feeds and speeds set, you now have to select the region (2D profile of the geometry).
- Go to Select Regions and use single select to select the 2D profile



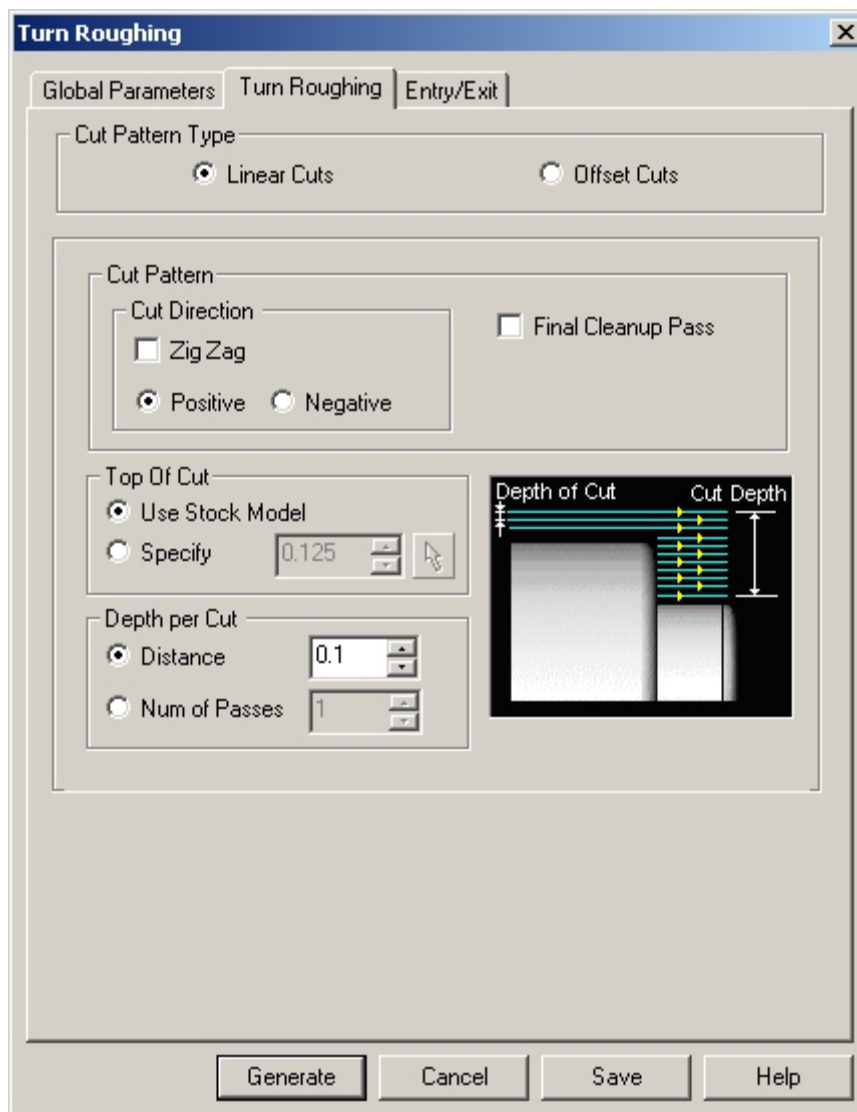
- Select **Turning / Roughing**, or click the **Machining** icon on the **Mops** tab of the Browser, and then select **Turning / Roughing**.



4. The **Roughing** window opens, in which you can set parameters for the toolpath.
5. In the **Global Parameters** tab, set the Approach type to **Inner Diameter**, the **Stock** value to set to 0.01 inches. This value defines the thickness of the layer that will remain on the part after the tool-path is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.



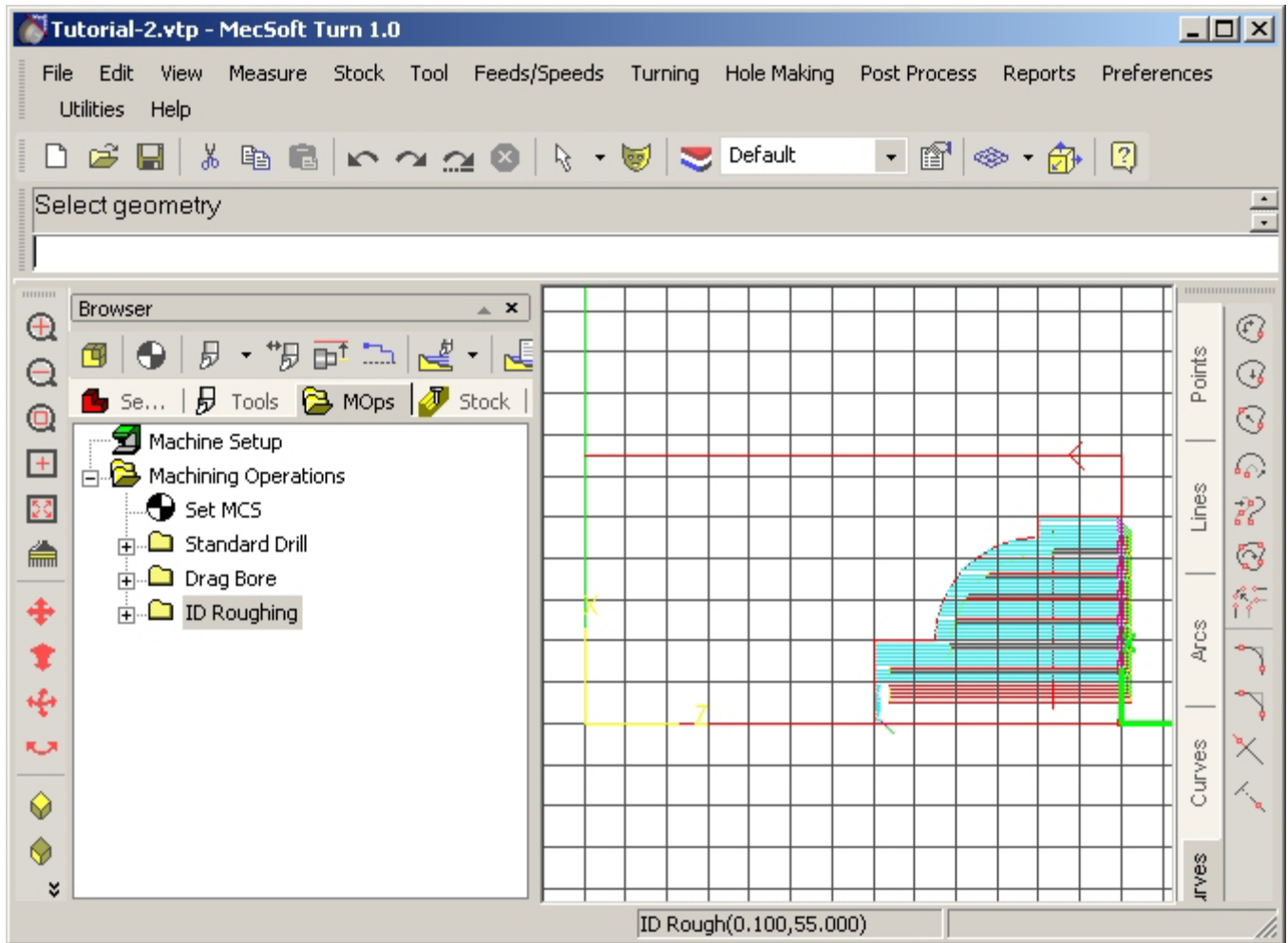
6. In the **Roughing Parameters** tab, set the **Cut Pattern Type** to **Linear**, uncheck **Final Cleanup Pass** and **Depth per Cut** to 0.1.



7. Leave the remaining parameters in the other tabs as they are, and click **Generate**, located at the bottom of the window.

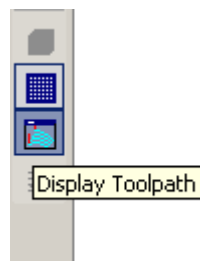
The window will disappear and an hourglass cursor will appear on the screen. When the computation is complete, the roughing toolpath will appear.

Getting Started with VisualTurn



Note: You can control the toolpath colors by selecting **Preferences / Color Preferences**.

If the toolpath is not displayed, make sure **Hide Toolpath** is not selected.

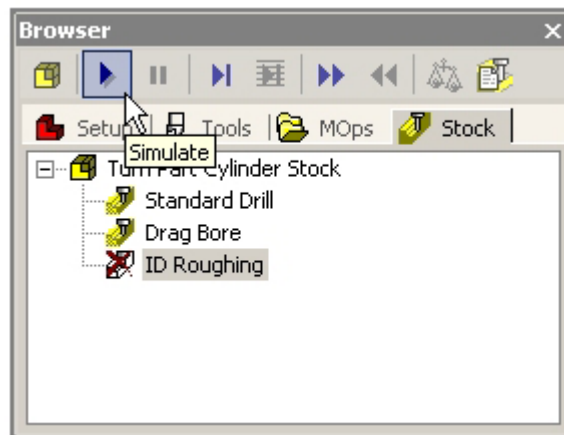


8. Right-Click on the Machine Operation name, i.e. **Turn Roughing** to edit it. Change it to **ID Roughing**.

Now that the first toolpath has been created, you can simulate it.

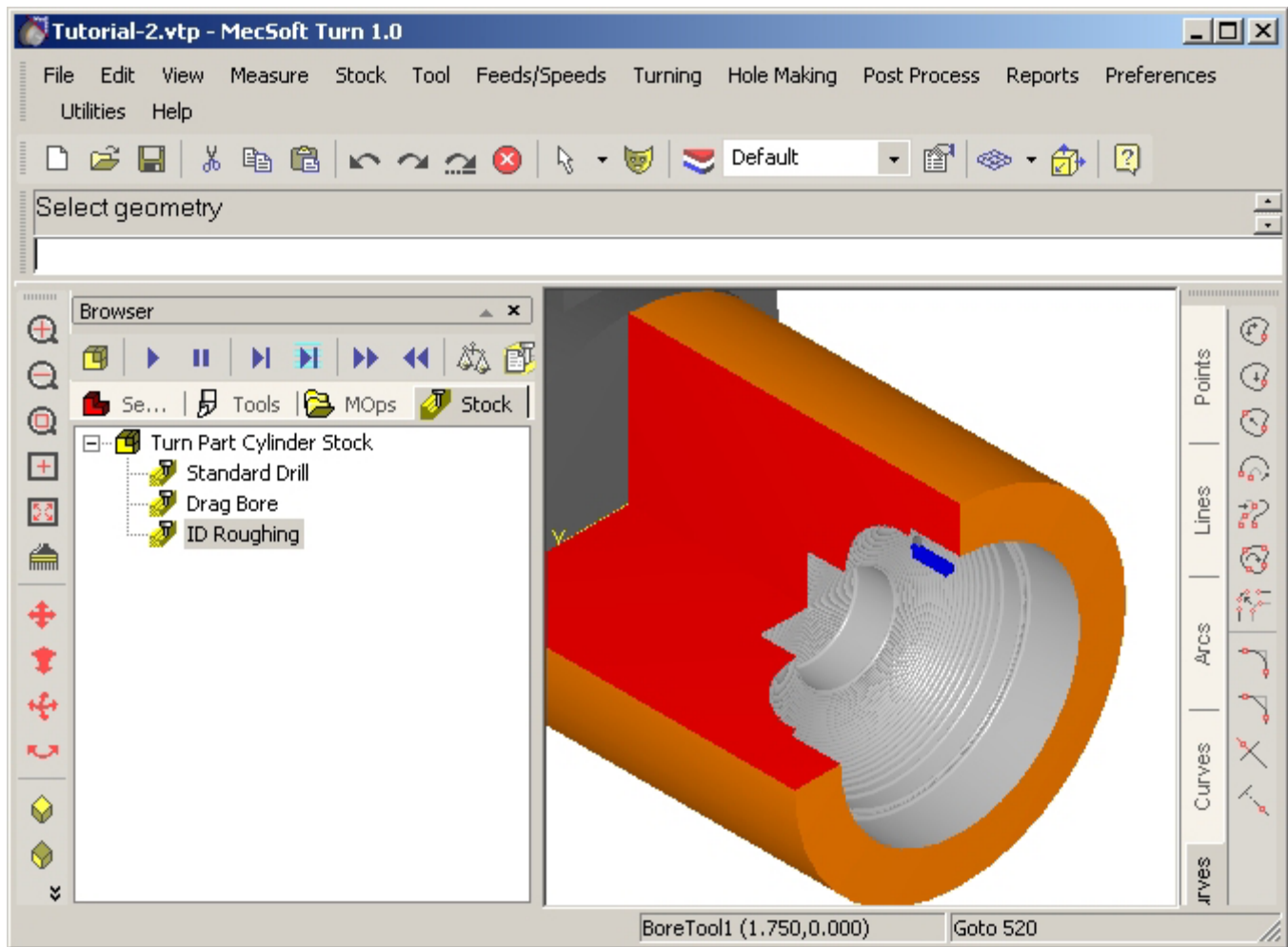
**To simulate the toolpath:**

1. To see how the stock looks after this toolpath, switch to the **Stock** tab. The cylinder stock box is displayed. The toolpath name is displayed with a red X, indicating that the simulation has not been run.
2. Select the ID Roughing toolpath and click **Simulate**.



Once the simulation is complete, the cut stock model will be displayed. This cut model can be used as input stock geometry for simulating the toolpath of subsequent machining operations.

Getting Started with VisualTurn



Note: You can also try clicking **Step** simulation to view a set number of tool motions at one time.

Tips: You can change the color of the stock and cut stock by selecting **Preferences / Color Preferences** and clicking the color box for **Cut Stock Color**.

The toolpath now has a “simulation complete” icon next to its name in the **Stock** tab.

Creating the Inner Diameter Finishing Toolpath

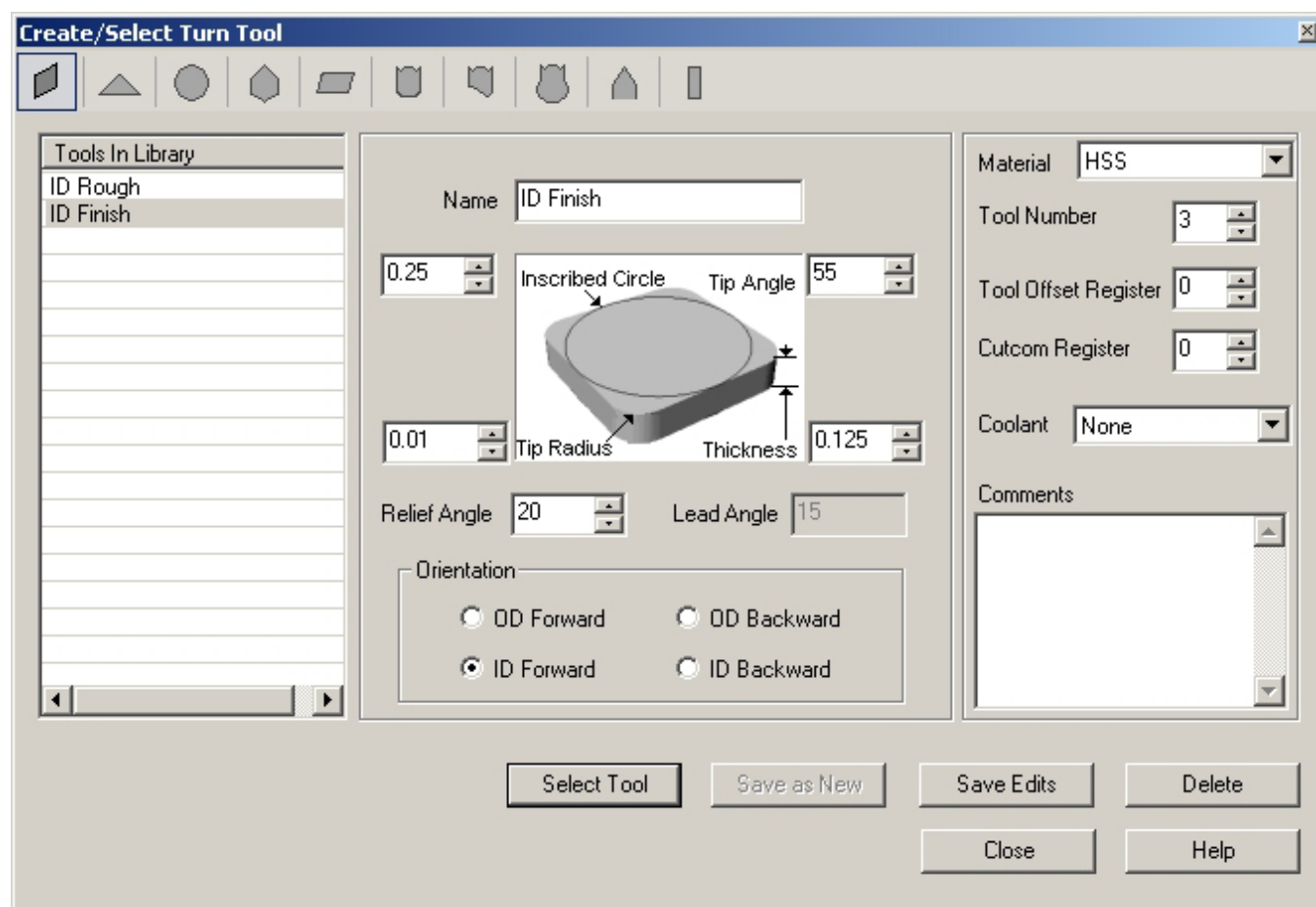
Once the roughing toolpath is generated, a finish toolpath can be created to remove the steps left by the roughing process, and to bring the stock equal to the part. In this exercise will use the **Finishing** method.



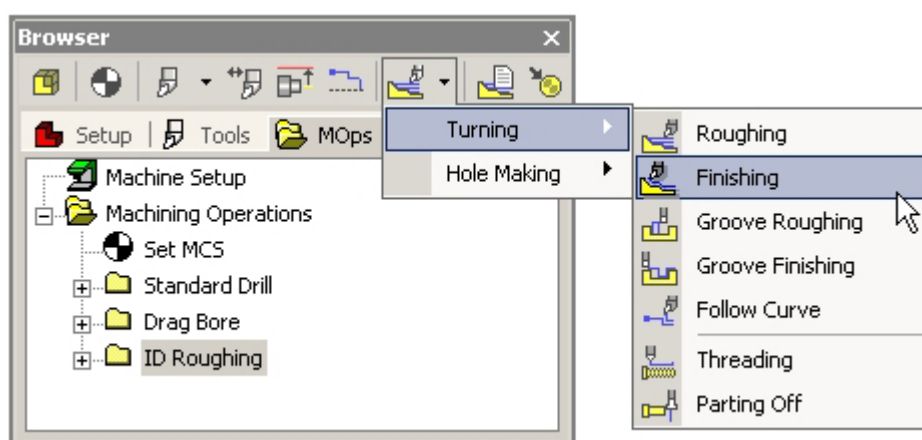
To create the Inner Diameter Finishing toolpath:

1. Return to the **Mops** tab.

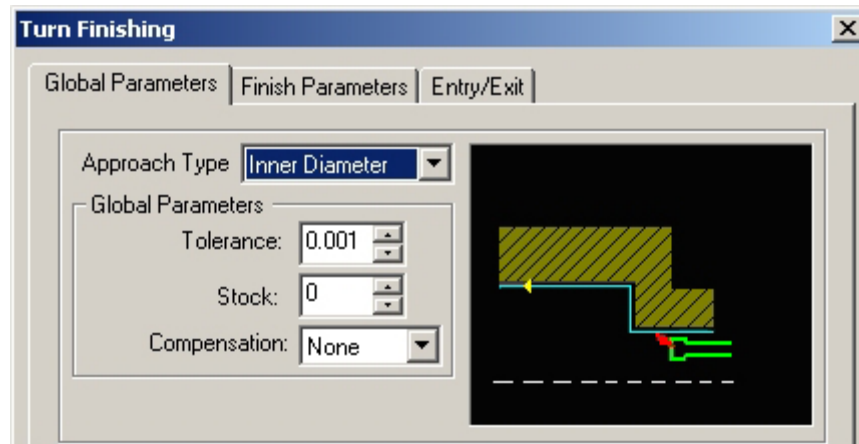
2. Activate the **Finish Tool**.



3. Select **Turning / Finishing**.



4. Set the Approach type to **Inner Diameter**

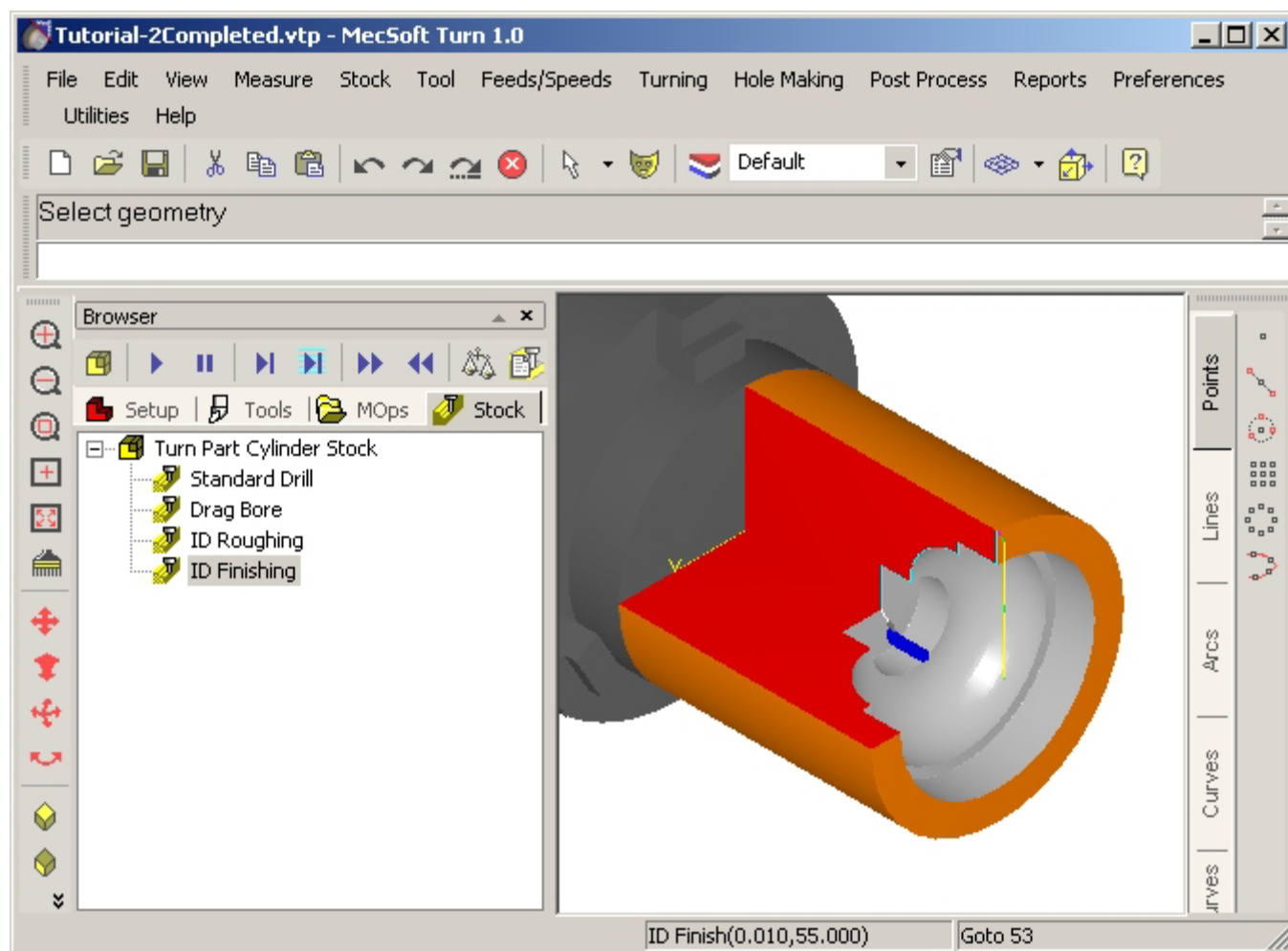


5. With the default set of values in the **Global and Finish Parameters** click **Generate**
6. Rename this operation **ID Finishing**.



To simulate the Inner diameter finishing toolpath:

1. Switch to the **Stock** tab. Select ID Finishing and click to Simulate.
2. Click **Simulate**.



To create the post-processed output:

In this exercise we will post-process all of the toolpaths at once.

1. In the **Mops** tab, right-click on the root folder (**Machining Operations**) to open the popup menu. Select **Post All**.
2. Browse to the desired output directory and assign a file name for the output. The default extension is *.nc. Then double-click on the post-processor you want to use (such as **Fanuc 1**).

End of Tutorial 2!

Tutorial-3 Grooving, Threading and Part off

In this tutorial, you learn to create Groove roughing, finishing and parting off operations

The stepped instructions are accompanied by explanatory and introductory text. Reading this text will help you understand the tutorial methodology and provide information about additional options available. However, if you prefer to work straight through the steps without any additional reading, look for the following symbol:



Don't forget to save your work periodically! You may want to save the file under a different name so that the original file will be preserved.

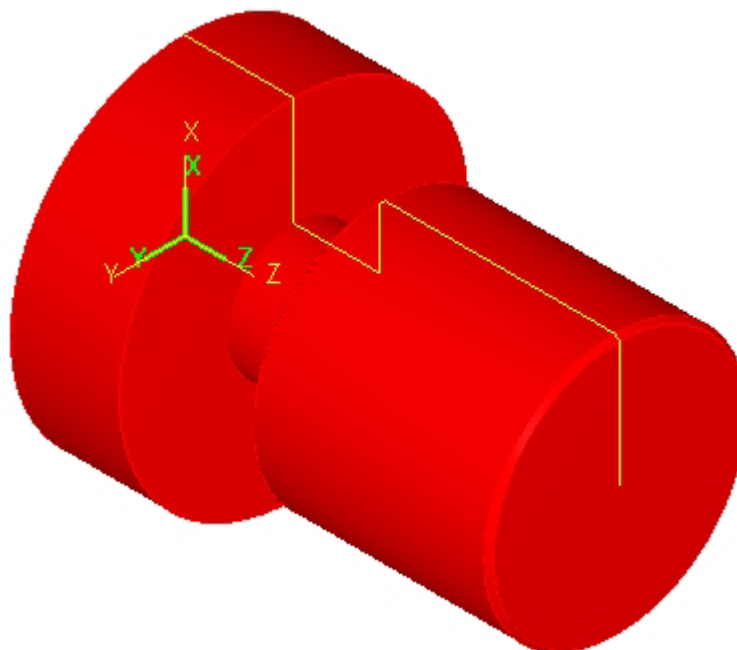
Creating the OD Roughing Toolpath

We will first create a OD roughing operation to remove most of the material from the stock.



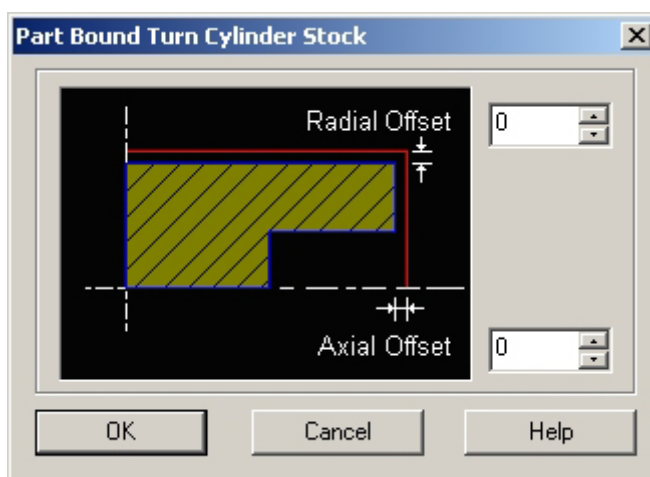
Steps for creating the OD Roughing toolpath

1. Select **File / Open**, or click the **Open Part File** icon from the **Standard** bar. Select the **Tutorial-3.vtp** file from the **Tutorials** folder.
2. From the setup tab click on the  **Slice 3D Part** on the Setup bar of the Browser. This generates a 2D profile of the 3D geometry.
3. Now select the 2D profile and click  **Resolve Part Region**. This resolves the 2D profile in the First Quadrant of the lathe coordinate system. (Positive X and Z axis)



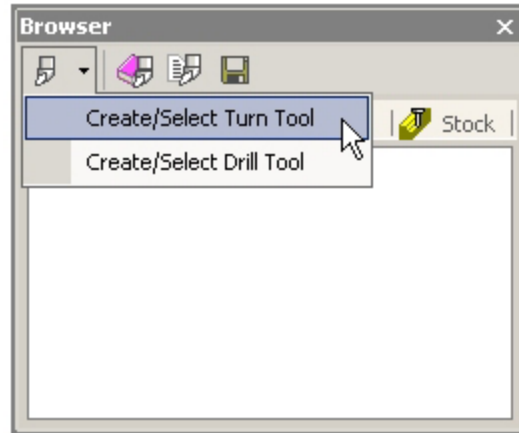
You may now save the file and start creating VisualTurn machining operations.

4. **Stock / Part Cylinder Stock**, or click **Create/Load Stock** from the **Setup** tab of the Browser and select **Part Cylinder Stock**.
5. Set **Axial** and **Radial offset** to **0**.



Select the tool to create the operation

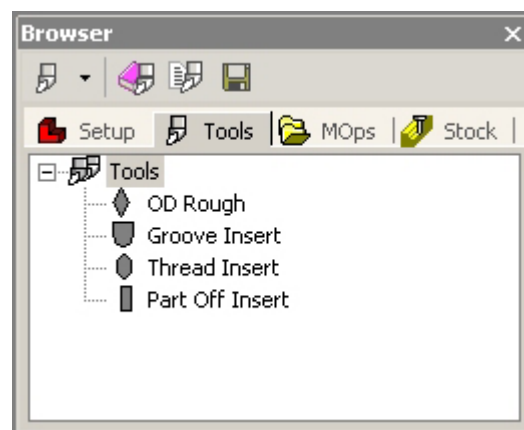
1. Select **Tool / Create/Select Turn Tool**, or click the **Create/Select Turn Tool** icon on the **Tools** tab of the Browser.



2. Create the tools with the following parameters
 - a. **Diamond Insert** – Name: OD Rough, Inscribed Circle: 0.25”, Tip Radius: 0.01”, Orientation: OD Forward
 - b. **Groove Insert** – Name: Groove Insert, Total Length: 1.5”, Length: 1.25, Tip Radius: 0.0625, Program Point: Left
 - c. **Thread Insert** – Name: Thread Insert, Length: 0.5”, Tip Radius: 0”, Nose Angle: 60 deg, Width: 0.26”, Thickness 0.125”
 - d. **Part off Insert** – Name: Part off Insert, Length: 2”, Width: 0.125”, Thickness 0.125”

Click **save as new tool** when you create a new tool.

3. Now that all the tools have been created click **close** to exit the create/select tool dialog. The tools tab should list the created tools as shown below



Note: You can double-click on any tool to open its definition window. This is an easy way to make changes, if needed.



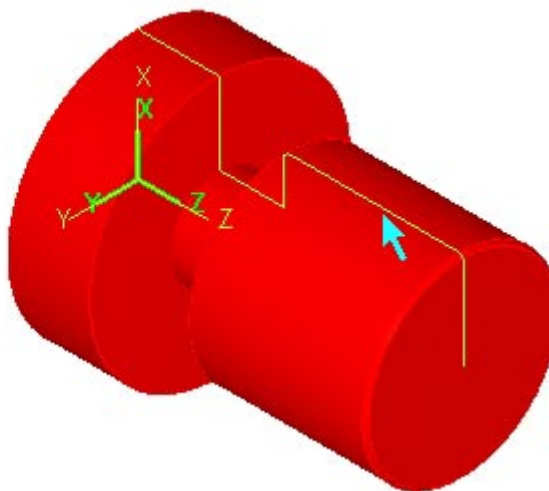
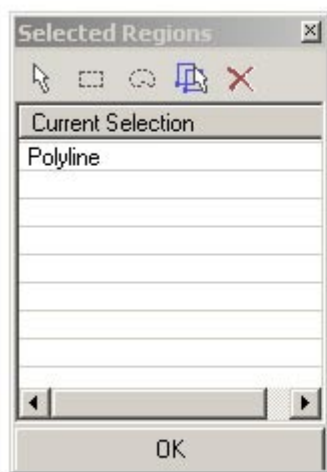
Set the Machine Co-ordinate System

1. Click on the **Set MCS** icon in the **Mops** tab of the Browser Bar or Double-Click on the Set MCS under Machining Operations
2. For this example, set the **MCS** origin to the **Stock Box** and Zero Face to **Right Most** face of the part. Click **OK**.
3. Under the Mops tab click on Create/Select Turn tool and tool select dialog pick the **OD Rough** tool.
4. We will leave the feeds and speeds with the default settings. Clearance Plane is set to automatic by default.



To create the Outer Diameter Roughing toolpath:

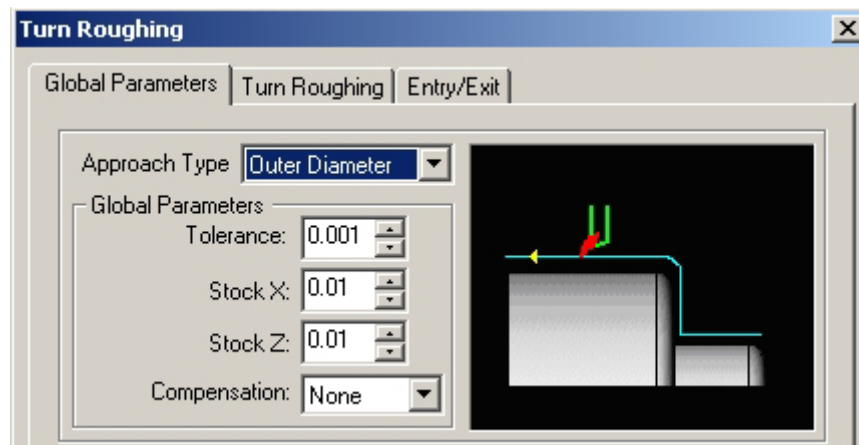
1. Once you have the tool selected and feeds and speeds set, you now have to select the region (2D profile of the geometry).
2. Go to Select Regions and use single select to select the 2D profile.



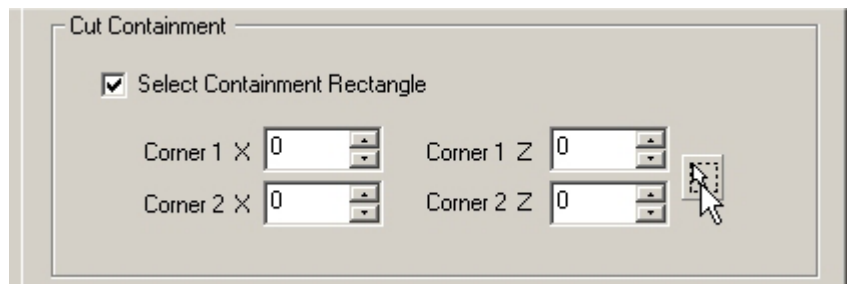
3. Select **Turning / Roughing**, or click the Machining icon on the **Mops** tab of the Browser, and then select **Roughing** operation. The Roughing window opens, in which you can set parameters for the toolpath.
4. In the **Global Parameters** tab, set the Approach type to **Outer Diameter**, the **Stock** value is set to 0.01 inches. This value defines the thickness of the layer that will remain on the part after the

Getting Started with VisualTurn

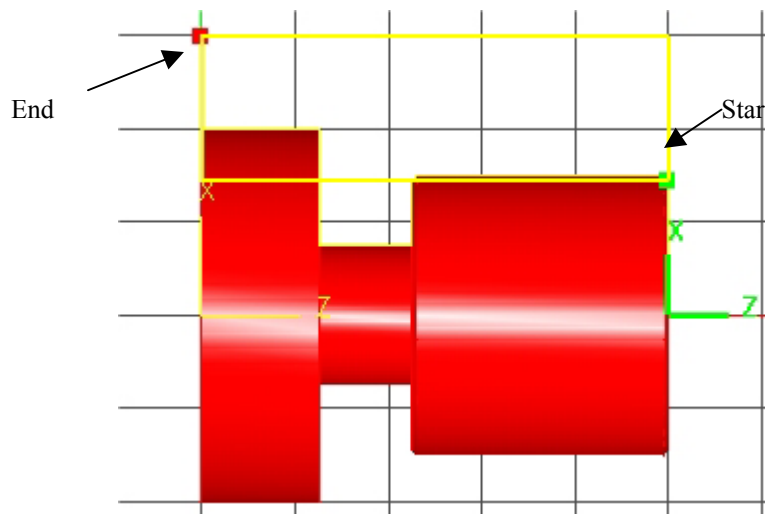
tool-path is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.



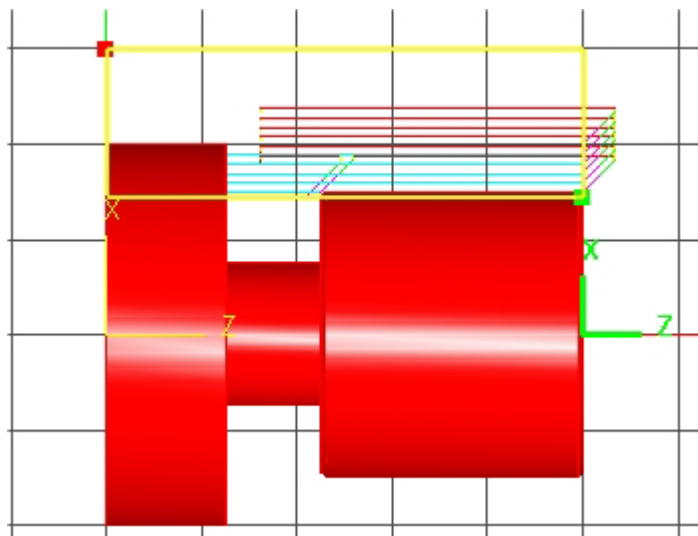
4. We will specify a containment region by setting Cut Containment Check at Start and End Points
5. With the Mouse select tool for Start and End select the start and End points as shown below.
6. Clicking the Mouse Select minimizes the Turn Finishing dialog



7. Select the containment as indicated below. Turn on end snap to pick the start and end points



8. In the **Roughing Parameters** tab, set the **Cut Pattern** to **Linear**, uncheck **Final Cleanup Pass** and **Depth per Cut** to 0.1.
9. Leave the remaining parameters in the other tabs as they are, and click **Generate**, located at the bottom of the window. When the computation is complete, the roughing toolpath will appear as shown below.



Note: VisualTurn checks for relief angle protection based on the tool geometry and part geometry.

10. Rename the Turn Roughing operation to **OD Roughing**
11. Switch to the Stock Tab, Highlight the **OD Roughing** toolpath and click **Simulate**.
12. Once the simulation is complete, the cut stock model will be displayed. This cut model can be used as input stock geometry for simulating the toolpath of subsequent machining operations.

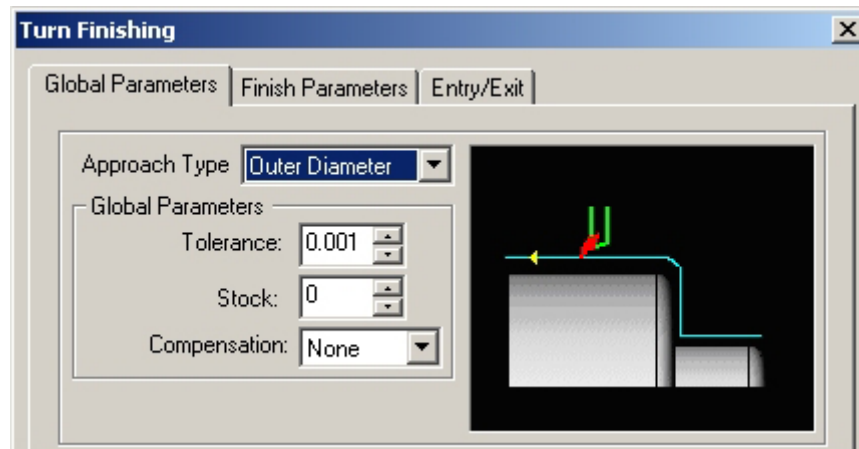
Creating the OD Finishing Toolpath

We will next create a OD finish operation to finish all accessible areas on the OD of the part.

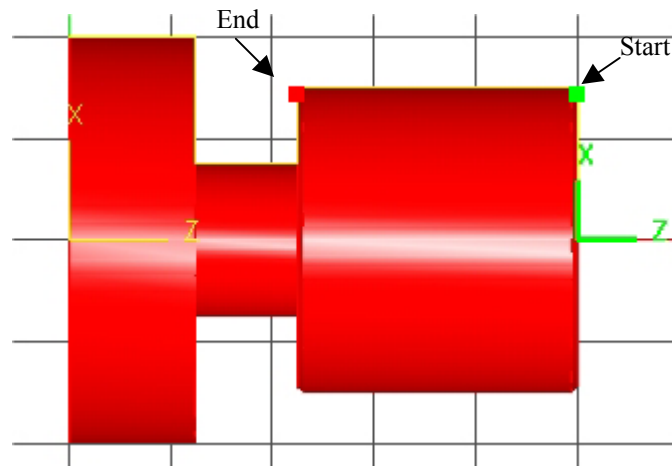


To Create OD Finishing Operation

1. Return to the **Mops** tab.
2. Select the **OD Rough** tool.
3. Select **Turning / Finishing**.
4. Set the Approach type to **Outer Diameter**



5. We will specify a containment region by setting Cut Containment Check at Start and End Points.
6. With the Mouse select tool for Start and End select the start and End points as shown below.



7. With the other parameters set to default we will now click generate to create the **OD Finishing** toolpath.
8. Rename the Mop to **OD Finishing**.
9. Switch to the **Stock** tab and select OD Finishing to Simulate.

Creating the Groove Roughing Toolpath

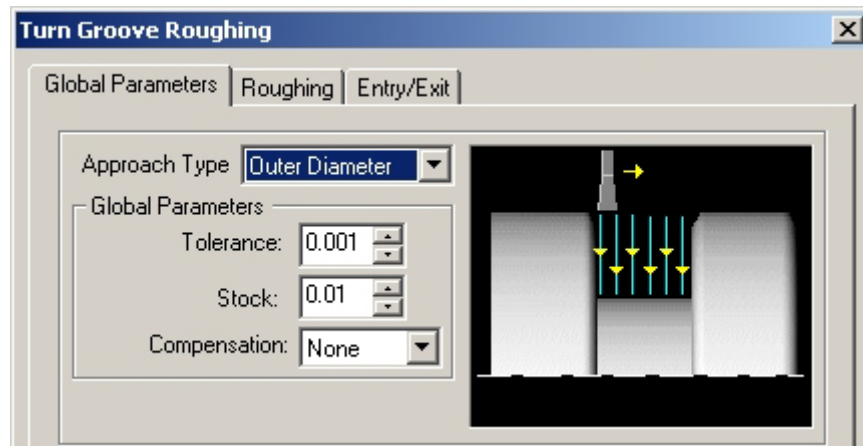
Next we will create a Groove roughing toolpath to rough out the groove feature on the part.



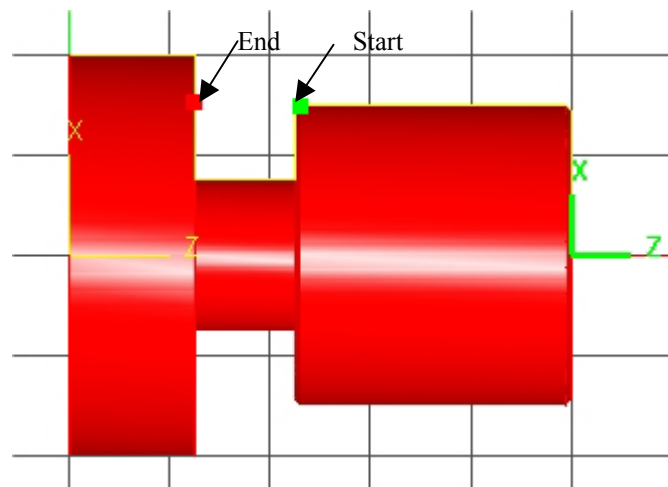
To create the Groove Roughing operation

1. Return to the **Mops** tab.

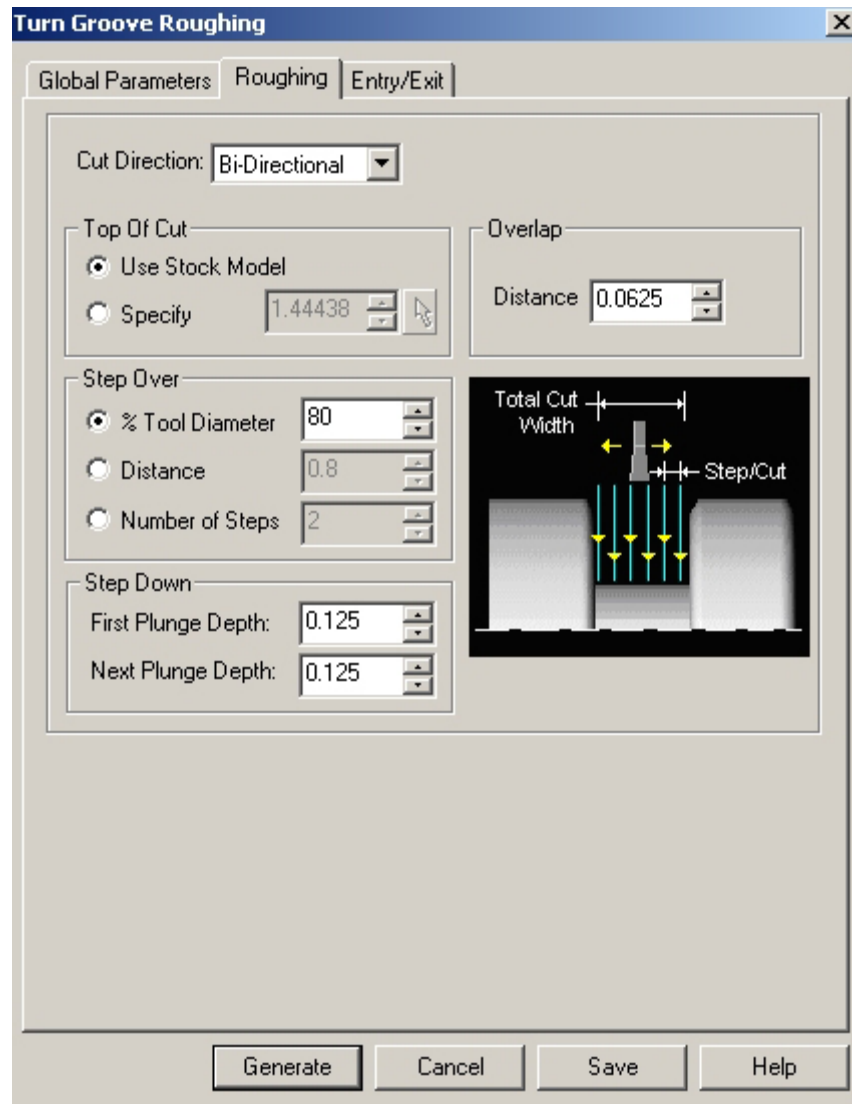
2. Select the **Groove Insert** tool.
3. Select **Turning / Groove Roughing**
4. Set the Approach type to **Outer Diameter**, **Stock** to **0.01**.



5. User must specify a containment region by setting Cut Containment Check at Start and End Points. Select the containment as shown below

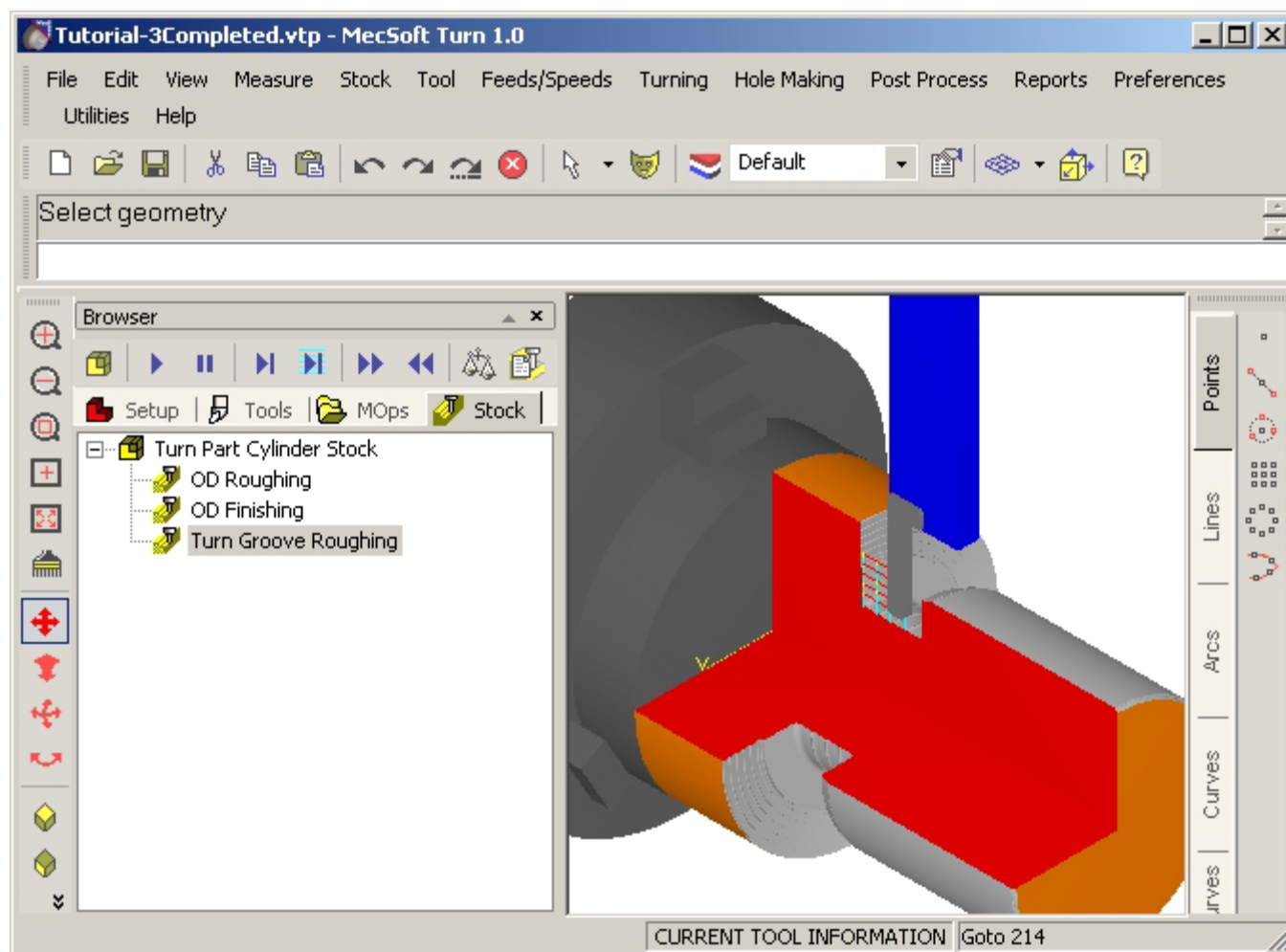


6. In the **Roughing** tab, set the **Cut Direction** to **Bi-Directional** and leave remaining parameters in the other tabs as they are, and click **Generate**, located at the bottom of the window



The window will disappear and an hourglass cursor will appear on the screen. When the computation is complete, the groove roughing toolpath will appear.

7. Switch to the stock tab, select **Turn Groove Roughing** and click  to Simulate



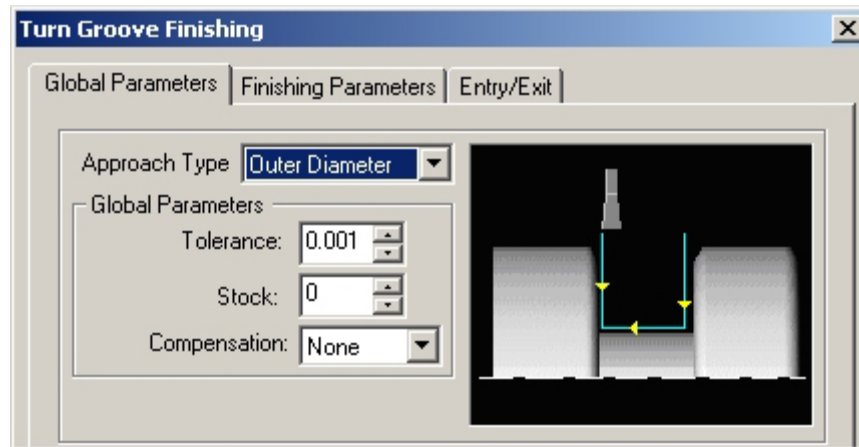
Creating the Groove Finishing Toolpath

Next we will create a Groove finishing toolpath to finish the groove feature on the part.

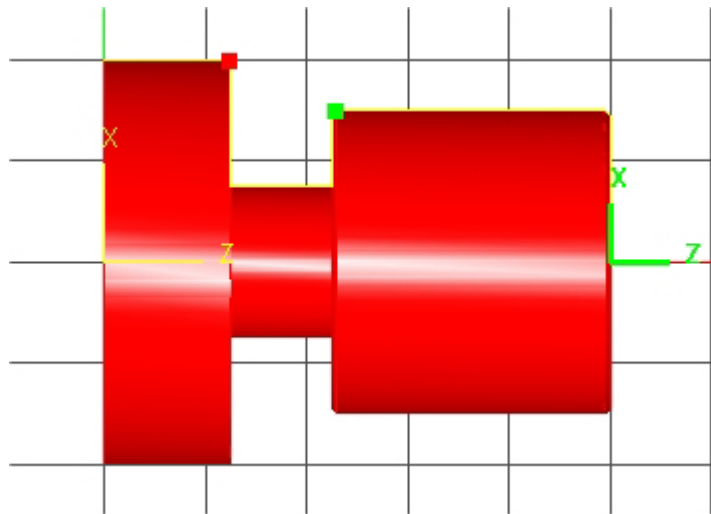


To create the Groove Finishing operation

1. Return to the **Mops** tab.
2. Select the **Groove Insert** tool.
3. Select **Turning / Groove Finishing**
4. Set the Approach type to **Outer Diameter**, Stock to **0**.



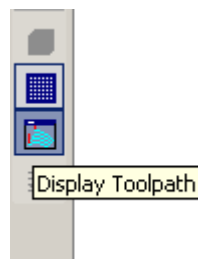
5. Specify a containment region by setting Cut Containment Check at Start and End Points. Select the containment as shown below



(Note: **Green** Indicates start point and **Red** indicated end point)

6. Leave remaining parameters in the other tabs as they are, and click **Generate**, located at the bottom of the window. Groove finishing toolpath will appear once the computation is complete.

If the toolpath is not displayed, make sure **Hide Toolpath** is not selected.



7. Switch to the stock tab, select **Turn Groove Finishing** and click  to Simulate

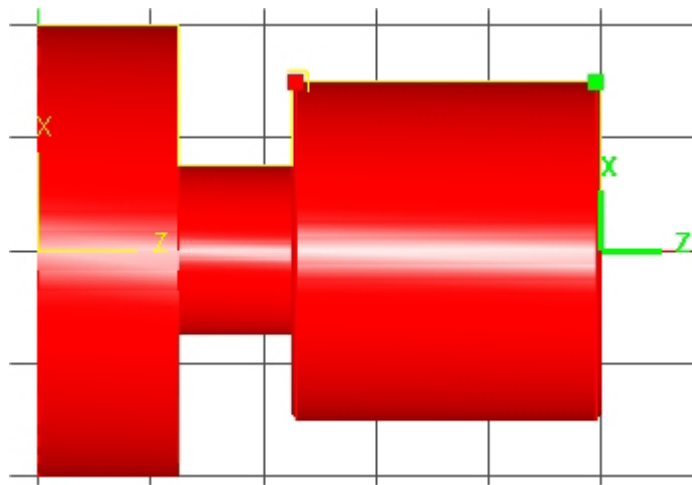
Creating the Threading Toolpath

We will next create the threads on the OD.



To create a Threading operation

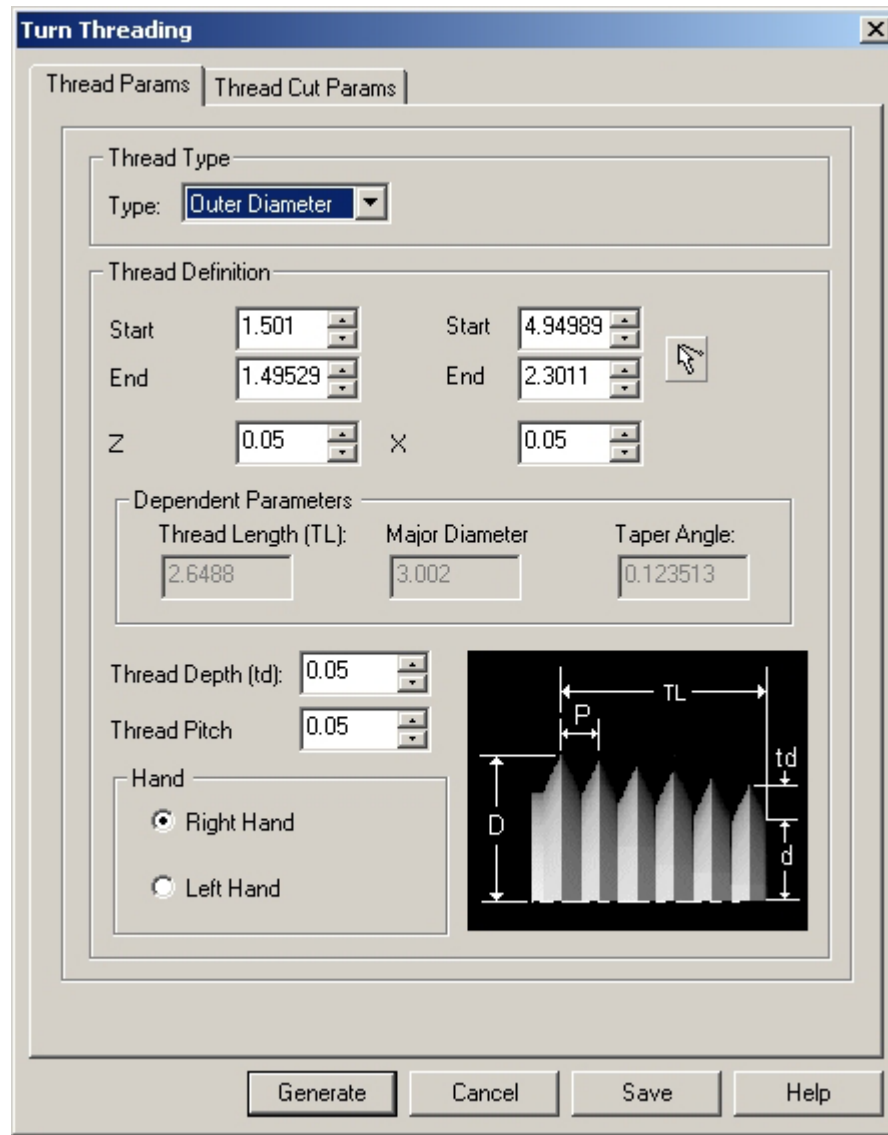
1. Return to the **Mops** tab.
2. Select the **Thread Insert** tool.
3. Select Turning / **Threading**
4. Set the Approach type to **Outer Diameter**
5. Specify a containment region by setting Cut Containment at Start and End Points. Select the containment as shown below



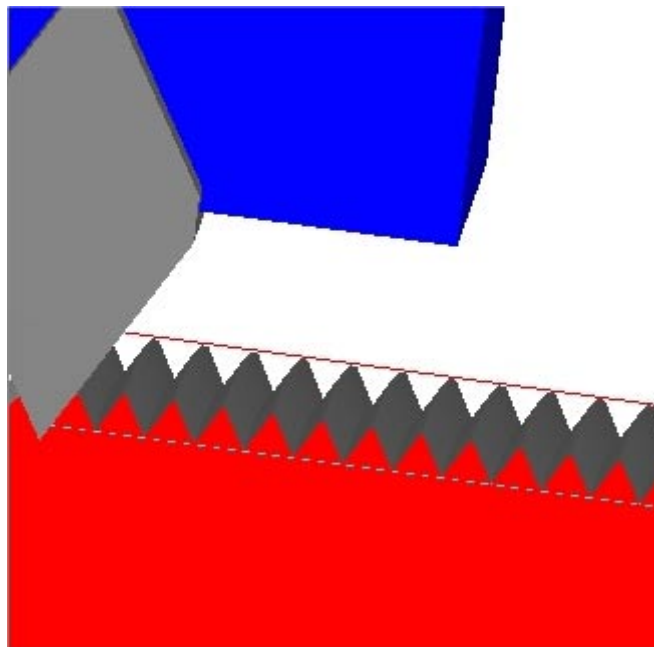
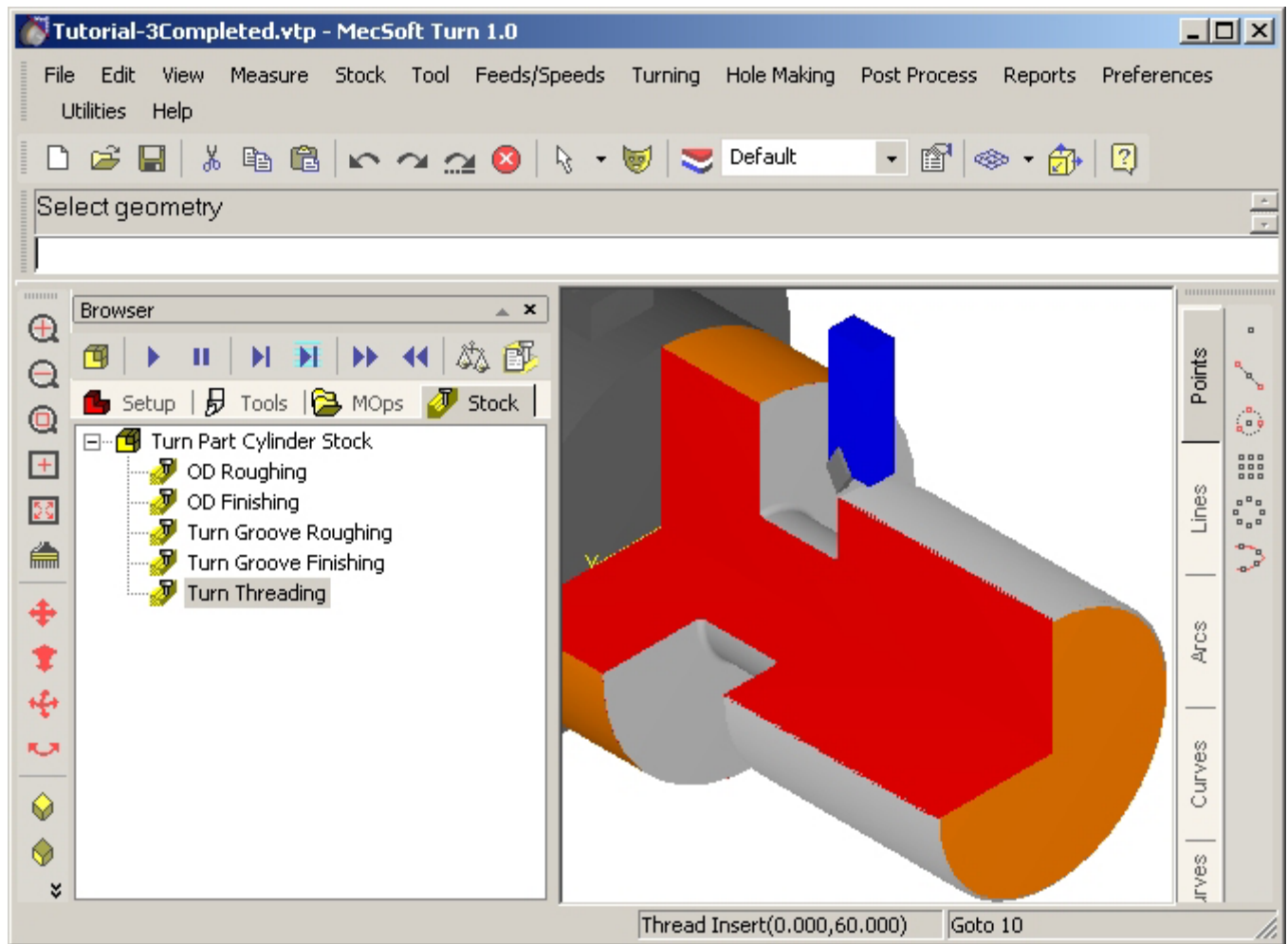
(Note: **Green** Indicates start point and **Red** indicated end point)

6. Set the **Thread Depth** to **0.05"**, **Thread pitch** to **0.05** and thread type to **Right Hand** Thread
7. Leave remaining parameters in the Thread Cut Params tabs as they are, and click **Generate**, located at the bottom of the window. Threading toolpath will appear once the computation is complete.

Note: Threading may take longer time to simulate when compared other turning operations as this involves data-intensive computation and rendering.



8. Switch to the stock tab, select **Turn Threading** and click  to Simulate



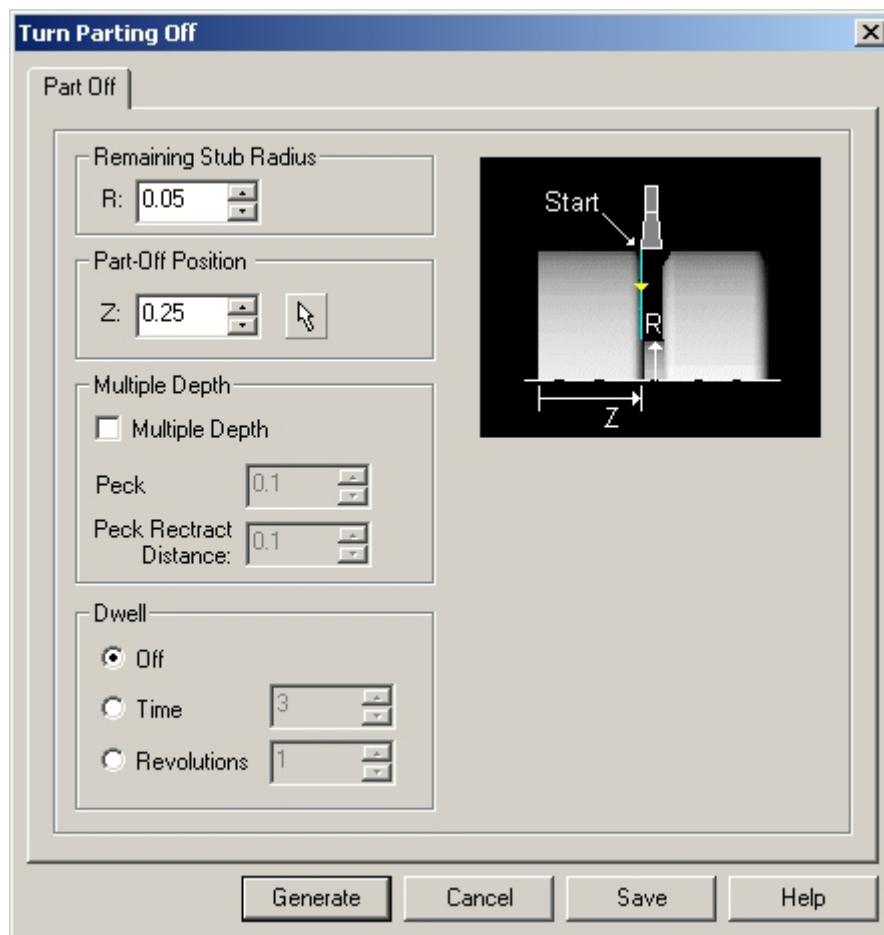
Creating the Parting-Off Toolpath

Finally we will create a parting-off operation to cut off the stock and remove it from the chuck.

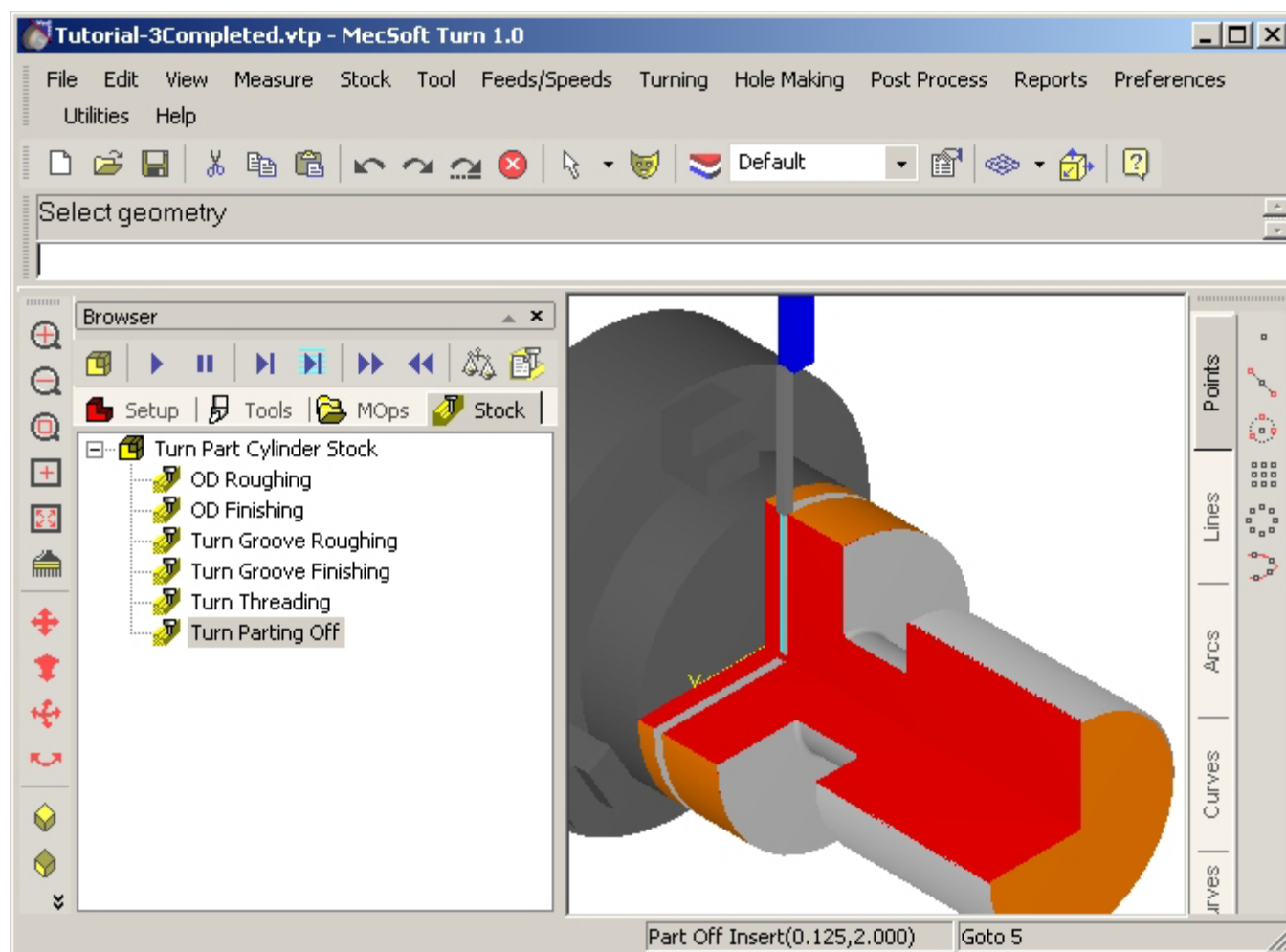


To create **Turn Parting-Off** operation

1. Return to the **Mops** tab.
2. Select the **Part off** tool.
3. Select Turning / **Parting Off**
4. Set the **Remaining Stub Radius** to **0.05"**, **Part-Off Position** to **0.25"**. Leave the remaining parameters to default click **Generate**, located at the bottom of the window.



5. Switch to the stock tab, select **Turn Parting-Off** and click  to Simulate



To create the post-processed output:

In this exercise we will post-process all of the toolpaths at once.

3. In the **Mops** tab, right-click on the root folder (**Machining Operations**) to open the popup menu. Select **Post All**.

Browse to the desired output directory and assign a file name for the output. The default extension is *.nc. Then double-click on the post-processor you want to use (such as **Fanuc 1**).

End of Tutorial 3!

Where to go for more help

In addition to the features described in this guide, VisualTurn has many more features designed to make it easier for you to create toolpaths and G-code. VisualTurn's complete on-line help provides reference information for each of VisualTurn's features and functions.

If you need additional help, or if you have any questions regarding VisualTurn, **first try the FAQ section** on our web site, www.mecsoft.com. Most of the common issues that users face are cataloged here. If you still have additional questions, visit our Users Forum at our web site to learn from other VisualTurn users. You can also contact us via e-mail at support@mecsoft.com.

Appendix I: Network Installation of VisualTurn

If you have purchased a network license of VisualTurn please follow the steps outlined below for the proper installation of the network enabled hardware key.

- 1) Install the **VisualTurn software** on the server machine as well as all the client machines connected to this server.
- 2) Install the **dongle drivers** on the server as well as all the client machines connected to this server. (Install Hardware Drivers should be on the VisualTurn CD).
- 3) If you do not find it download it from <http://www.safenet-inc.com/support/tech/sentinel.asp> and select Sentinel Super Pro Download and Run Sentinel Protection Installer v7.2.1
- 4) Install the Key server installation program: **RainbowServerInstaller.exe** on the server. You will find this in the VisualTurn install directory, typically: C:\Program Files\MecSoft Corporation\VisualTurn 1.0
- 5) Set an environment variable, VMILL_LICENSE_HOST on each of the client machines to the servers' IP Address. This can be done as follows:
 - a. Go to **Start->Control Panel->System**
 - b. From the System Properties dialog box that pops up select the **Advanced** tab.
 - c. Click on the **Environment Variables** button at the bottom.
 - d. In the Environment Variables dialog click on the **New** button under **System variables**
 - e. In the New System Variable dialog box that pops up, define

Variable Name = VMILL_LICENSE_HOST

Variable Value = IP Address of the server machine

Hit the **OK** button.
- 6) Now plug in the dongle (parallel/USB) to the port.

To work across different subnets, please do the following in addition to the above instructions, open the UDP port 6001 in any router installed on the network, this will allow the communication to go across.

Appendix II: Trouble shooting VisualTurn Installation

If you have followed the installation steps outlined in the installation section correctly and are unable to load and run VisualTurn correctly follow these troubleshooting steps to correct the problem.

Troubleshooting the Software Installation

Make sure that the software was correctly installed. To do this you can browse to the installation folder of VisualTurn and make sure that the file **VisualTurn1_0.exe** is present. Also make sure that all the folders described in the following section are correctly installed. If you detect an incorrect installation, un-install the software completely and re-install the software using the product CD again. You can un-install the software by selecting the **Add or Remove Programs** option under the **Control Panel** settings of your computer.

VisualTurn Installation Folder

VisualTurn installation creates a main installation folder whose name and location you can specify during the installation process (or accept the default location of C:\Program Files\MecSoft Corporation\VisualTurn 1.0). This folder contains the VisualTurn executable and *.dll files. There are also several subfolders in the installation directory:

Data: Contains tool library files - **DefaultEnglishTools.vtl** and **DefaultMetricTools.vtl**. These files can be used as they are, or you can use them as templates and customize them with your own data. You will also find a speeds/feeds & material library file called **VTFeedsSpeedsEng.txt** and **VTFeedsSpeedsMet.txt**. For more information on how to modify these tool library files, please refer to VisualTurn's online help.

Examples: Contains various example files that you can experiment with. There are files from other CAD systems you can import, as well as VisualTurn files (*.vtp). The *.vtp files contain saved machining operations that you can study and modify.

Help: Contains the online help files used with VisualTurn. You can open these files directly from this folder, or access them within VisualTurn.

Posts: Contains the standard set of post-processor (*.spm) files. Additional post-processor files can be obtained from MecSoft Corporation. If you receive additional *.spm files, be sure to place them in this folder, so that VisualTurn will recognize them.

Tutorials: Contains a tutorial and several part files to help first-time users get familiar with VisualTurn. These are similar to the tutorials presented in this guide, in onscreen format. To launch these tutorials, open the **VisualTurn1.0Tutorials.chm** file, and use the table of contents or arrows to browse through the steps.

FeaturePresentation: Contains all the files necessary to run a presentation of features of VisualTurn for first time users.

Troubleshooting the Hardware Security Key

If you have installed the dongle driver and connected the dongle but VisualTurn is not running properly, try restarting your computer. If that still does not work do the following:

For Users with USB Dongle (Hardware Key)

1. Close VisualTurn and remove the USB dongle.
2. Go to <http://www.safenet-inc.com/support/tech/sentinel.asp> and select Sentinel Super Pro
3. Download and Run the SSD Cleanup v1.1
4. Restart your computer
5. Go to <http://www.safenet-inc.com/support/tech/sentinel.asp> and select Sentinel Super Pro
6. Download and run Sentinel Protection Installer v7.2.1
7. Plug the dongle back in and launch VisualTurn 1.0

For Users with Parallel Dongle (Hardware Key)

1. Close VisualTurn.
2. Go to <http://www.safenet-inc.com/support/tech/sentinel.asp> and select Sentinel Super Pro
3. Download and Run the SSD Cleanup v1.1
4. Restart your computer
5. Go to <http://www.safenet-inc.com/support/tech/sentinel.asp> and select Sentinel Super Pro
6. Download and run Sentinel Protection Installer v7.2.1
7. Launch VisualTurn 1.0

If the above method does not work, download the **Sentinel Medic** from the Rainbow website (<http://www.safenet-inc.com/support/tech/sentinel.asp> and select Sentinel Super Pro). Install it and go to Start->Programs->Rainbow Technologies->Sentinel Medic. Click **Find SuperPro** and send the following information that appears on the screen to support@mecsoft.com, so that we can identify and fix your specific problem:

1. System Driver Info
2. Status
3. Description
4. Medic Says

Getting Started with VisualTurn

Troubleshooting VisualTurn Display

If you are experiencing problems with the way VisualTurn appears on the screen, try the following:

For Windows ME, 2000 and XP:

1. Right-click anywhere on the desktop and select **Properties** from the menu.
8. Open the **Settings** tab and click **Advanced**.
9. Open the **Troubleshoot** or **Performance** tab and set **Hardware acceleration** to **none**.

If you are still having problems, reinstall the video drivers of your video card. Or you can try another video card to see if the problem is specific to your card.

If VisualTurn opens as a minimized window and closes when maximized (this happens on rare occasions, typically on computers with defective display cards), it is probably due to bad window coordinates stored in your computer's registry. Try the following to eliminate this problem:

1. Press Windows + R button.
2. Type in **regedit** and click OK.
3. In HKEY_CURRENT_USER / Software, delete the **VisualTurn1.0** entry.

Appendix III: Description of the Browser toolbar buttons

This appendix provides a short description of each of the toolbar buttons in the browser window.

Setup Tab Toolbar

The Setup manager displays the three types of geometry that can be created and manipulated in VisualTurn: **Surfaces/Meshes**, **Curves** and **Stock**

The first icon represents the **Part**. For an imported part, the full path is indicated. If the part consists of surfaces, each surface is represented as a **Mesh**. You can click on each mesh name to highlight its corresponding surface.

Curves in the model are regions used to define machining boundaries. If you have created machining operations, the toolpath for the respective curve will appear underneath it.

Lastly, the **Stock** icon indicates the type of stock. You can double-click or right-click to create a different type of stock, or delete the stock or export it to an *.stl file. A red star next to this icon indicates that the work-in-progress stock model corresponding to this operation needs to be created.



Machine Setup: Define the machine tool by specifying its tool change position and travel limits



Set Post Options: Set the path for the post processor files and program to be used for displaying the posted file



Convert XY to ZX: Convert part geometry from normal XY to ZX (lathe coordinate system)



Slice a 3D Part: Use this to create a 2D profile of the 3D geometry



Resolve Part Region: Resolves the 2D profile into First quadrant of the lathe coordinate system (X and Z axes)



Create/Load Stock: Create/load a stock material from the list of stock types



CAM Preferences: Set up Machining and Color Preferences



CAM Utilities: Access to Post Processor Generator, DNC to Machine, G Code Editor and MCU.

Tools Tab Toolbar

This tab lists all tools currently defined in the file. If you have created machining operations, the toolpath will appear in the **Tools** tab underneath the tool it uses. You can rename and delete tools, but you cannot delete a tool that is used in a toolpath. Double-click a tool icon to edit its parameters.



Create Turn/Drill Tool: Launches the Tool Creating dialog to create Turn/Drill tools



Load Tool Library: Load tools from External Library.



Tools Info: Lists information on all tools



Save Tool Library: Save current tools to library

Mops Tab Toolbar

“Mops” stands for Machining Operations. All toolpaths you create are listed here, in order of creation. Within each toolpath folder you can edit its various components, such as tool, regions, or cut parameters, by double-clicking the relevant icon. Right-clicking on a toolpath name provides several options, including simulation, generation, and post-processing.

If you make any changes to a toolpath’s parameters, the yellow folder icon for that toolpath will turn red. This indicates that the toolpath needs to be regenerated.



Create/Load Stock: Create/load a stock material from the list of stock types



Set MCS: Set the origin of the machine coordinate system, with respect to part, or at an exact location.



Create/Select Tool: Opens a window in which you can define all turning and drilling tools that will be needed in the machining operations.



Set Feeds/Speeds: Defines the feed and speed rates for cutting, rapid, approach, engage, retract, and depart tool motions.



Clearance Control: Sets the level away from the part for safe rapid tool motion.



Select Regions: Provides several methods for selecting curves that will act as machining boundaries.



Machining Methods: Choose the type of toolpath you want to create. Machining methods are described in the next section.



Machining Operations Info: Displays information like machining operation name, cut feed, machining time for each machining operation.



Post Process: Sends the toolpath code to the machine.

Stock Tab Toolbar

The commands on the stock tab toolbar are used for toolpath simulation. For simulation to work, you must have stock geometry defined, and the stock must be displayed. The VisualTurn simulator enables you to view your toolpath in action, reflecting what the actual model would look like after machining. Simulation can also be used to catch errors. The cut stock can also be visually compared with the part model to indicate any areas of uncut or over-cut material.



Create/Load Stock: As you've already seen, this tool is used to create various types of stock material. Generally a stock is created automatically, depending on the type of toolpath you create, but you can always change the stock.



Simulate: The simulation will be run for the entire toolpath, and the end result of the material removal will be displayed.



Pauses: Stops the simulation.



Step: The simulation will be performed for a specified number of toolpath motions. To set the step value, open the **Simulation Preferences** (the last icon on the toolbar), and adjust the **Maximum Display Interval**.



Step Z Levels: Shows the resultant stock after each Z level. This feature is disabled in VisualTurn.



Simulate to End: Jumps to the end of the simulation.



Rewind: Jumps to the start of the simulation.



Compare Part/Stock: Performs a visual comparison of the stock material against the part model. You can color-code areas based on the amount of material remaining or overcut.



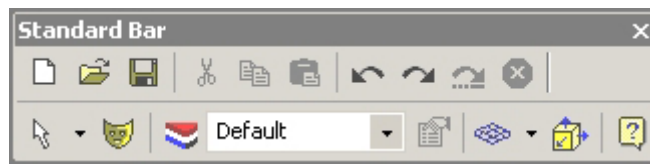
Simulation Settings: Opens the **Simulation Preferences**, in which you can set various properties of the simulation and display.

Appendix IV: Description of other toolbar buttons

This appendix provides a short description of each of the buttons found in various toolbars of VisualTurn other than the ones in the browser. For the description of the latter please refer to previous chapter.

The Standard Bar

Before beginning, the first commands you should know are on the first few on the **Standard Bar**. These commands are used to load and save files, and can also be accessed from the **File** menu.



New: Creates a new file.



Open: Loads part geometry into VisualTurn. This geometry is typically imported from other CAD formats, but can be created from within VisualTurn as well.



Save: Saves the current file as a *.vtp file. We recommend saving your work periodically, to avoid losing data.



Cut Selection: Removes the current Selection.



Copy Selection: Copies the current Selection.



Paste Selection: Pastes the copied Selection.



Undo: Undo Previous Command



Redo Previous Command: Pastes the copied Selection.



Command Recall: Recalls Previous Command.



Stop: Stops the current operation



Select: Provides Selection choices (See Select Regions for details).



Selection Mask: Customize selection choices



Layer Manager: Controls Layer access and properties.



Layer: Displays active layer.



Properties: Properties of selected objects.



CPlane: Change Construction plane orientation/settings.



CSYS Manager: Coordinate System Manager (MCS and WCS)



Help Topics: Lists Help Topics.

View Bar

The **View** bar is used for view and display manipulation. By default, it appears vertically along the left side of the screen, but you can dock it anywhere.



Each of the view functions is described below:



Zoom In: Doubles the displayed size.



Zoom Out: Halves the displayed size.



Zoom Box: Zooms in on an area you specify by defining a rubber-banded rectangle.



Center View: Centers the view about a selected point.



Fit View: Fits the entire part into display extents.



Repaint View: Repaints, or refreshes, the view.



Dynamic Pan View: Pans the view by holding and dragging the mouse.



Dynamic Zoom View: Zooms the view by holding and dragging the mouse. Move the mouse up to zoom in, move the mouse down to zoom out.

Getting Started with VisualTurn



Dynamic Rotate View: Rotates the view by holding and dragging the mouse. The rotation follows the mouse movements as if there were an imaginary trackball at the center of the view.



Dynamic Rotate View About Z: Rotates the view about the Z-axis and the origin point



Top View: Displays the top view - the ZX plane.



Right View: Displays the right view - the YZ plane.



Front View: Displays the left view - the XY plane.



Iso View: Displays the model in isometric projection.



View to CPlane: Sets the view so that the construction plane is parallel to the screen.



Shade Part: Toggles the display of part geometry between shaded and wireframe modes.



Hide Stock: Toggles the display of the stock geometry.



Display Grid: Toggles the display of the construction grid.



Hide Toolpath: Toggles the display of the toolpath associated with the current machining operation.



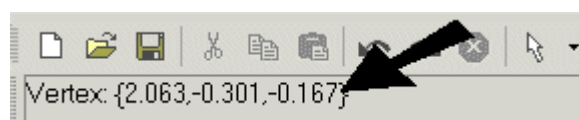
Display Next Z: Displays the toolpath for each level. This button is disabled for VisualTurn.

Measurement Bar

One of the steps in the next part of this tutorial is to measure the part. Measuring tools can be found on the **Measure** menu and on the **Measurement bar (View / Toolbars / Measurement)**.



When a measurement is calculated, it is displayed in the bar at the top of the screen.



Vertex Coordinates: Displays XYZ coordinates of a selected point.



Measure Distance: Measures the distance between two points. The point coordinates and the distance between them will be displayed.



Measure 3 Vertex Radius: Measures the radius of an arc spanning 3 points. The point coordinates and their arc radius will be displayed, and the arc will appear temporarily.



Measure Arc Radius: Measures the radius of an arc or circle.



Part Bounding Box: Calculates the dimensions of the bounding box around the part.



Part Center: Calculates the coordinates of the center of the part.

Status Bar

The **Status Bar**, located at the bottom of the screen, is used to display information about the current activities.



The left-most field displays the current command and any prompts or help information associated with this command. If you place the cursor over an icon, its description will appear here.

The next field indicates the active tool, if any. The name of the tool, followed by the diameter and corner radius, is displayed.

The next field indicates the progression of toolpath simulation (“Goto”), displaying the number of the motion being simulated. When the simulation is complete, the last motion number will be displayed.

The icon fields are snaps that can be toggled on and off:



Grid Snap (snaps to grid points)



Ortho Snap (constrains lines to be horizontal or vertical)



Origin Point Snap (snaps to the origin)



End Point Snap



Mid Point Snap



Center Point Snap



Intersection Point Snap

Getting Started with VisualTurn

 **Quad Point Snap** (snaps to 0, 90, 180, and 270-degree points of circles and arcs)

The second to last field displays the work units – inches or mm.

The last field shows the current location of the cursor as X, Y, Z coordinates. These values update as the cursor moves.

Geometry Bar

Geometry bar comprises of 6 tool tabs, each of which is described below. Each tool in the first four categories (Points, Lines, Arcs, and Curves) is described below. (Toolbars are shown rotated 90 degrees.)

Points



 **Point:** Creates a point by selecting it on screen or entering its coordinates. Points are used as references for other region tools.

 **Mid Point:** To create a point at the midpoint of a line, click this icon and select the endpoints of the line.

 **Center Point:** To create a point at the center of a circle, click this icon and then select three points of the circle.

 **Point Grid:** Specify the number of spaces between points in U and V, then select the opposite corners of the grid.

 **Bolt Circle:** Creates a circular array of points.

 **Points on Curve:** Creates a specified number of points evenly spaced along a selected line or curve.

Lines



 **Line Segment:** Creates a line by selecting two points.



Polygon/Polyline: Select the vertices of the polygon. If you want to close the region, move the cursor close to the start point of the region and select it. Right-click to finish.



Rectangle: Creates a rectangle by selecting its opposite corners.



Rounded Rectangle: Creates a filleted rectangle; first select the opposite point then set the rounding radius.



Line at Angle: Creates a line at a set angle from a specified baseline.



Line from Mid Point: Creates a line that extends the same distance on either side of a selected point.



Tangent Line: Creates a line tangent to a curve or collinear to an existing line.



Normal Line: Creates a line perpendicular to a curve or line.



Line Tangent to 2 Curves: Creates a line tangent to two curves or a curve and a line.



Line Normal to 2 Curves: Creates a line normal to two curves or a curve and a line.



Line Tangent and Normal: Creates a line tangent to one object, and normal to another.

Arcs



Circle Center, Radius: Creates a circle by selecting its center and a point on the circumference.



Circle Start, Diameter: Creates a circle by selecting two opposite points (diameter endpoints).



3 Point Circle: Creates a circle by selecting three points on its circumference.



Circle Tangent to 3: Creates a circle tangent to three objects.



Arc by Center, Start and Angle: Creates an arc by selecting the arc center, start point, and end point.



Arc: Start, End and Point: Creates an arc by selecting its endpoints, then setting its radius.

Getting Started with VisualTurn



3 Point Arc: Creates an arc by selecting the start point, a point on its circumference, and the endpoint.

Curves



Text: Creates a text string.



Spiral: Creates a flat coil.



Helix: Creates a 3D coil.



Single Flat Area Region: Creates a region bounding a single flat area.



All Flat Area Region: Creates bounding regions around each flat area.



Extract Edge Curves: Creates a region along a chain of edges. This is useful for creating a region along the outer boundaries of a surface. Select one edge, and all edges in the chain are automatically selected.



Bounding Region: Creates a rectangular region along the XY plane of the part's bounding box.